Section Builder: User's Manual

Olivier A. Bauchau
School of Aerospace Engineering,
Georgia Institute of Technology
Atlanta, GA, USA.

February 6, 2007

Contents

Chapter 1

Introduction

1.1 Overview of *SectionBuilder*

SectionBuilder analyzes beam cross-sections. The analysis proceeds in four steps. The menu of SectionBuilder, depicted in fig. 1.1, reflects these four steps.

1. First, the configuration of the cross-section is defined, this step is discussed in section 1.2. The two-dimensional geometric configuration of the section must be defined, together with the physical properties of the materials it is made of. Two avenues are available for this step. First, *parametric shapes* can be used, such as I-sections, C-sections, or a variety of commonly used cross-section. Second, more *complex sections of arbitrary configuration* can be constructed. The process of defining the section involves a number a dialog windows described in chapter 2 for the parametric shapes and in chapter 3 for complex sections of arbitrary configuration.

2. Second, a finite element mesh of this two-dimensional problem is created, as discussed in section 1.3. Clicking the first icon of the menu shown in fig. 1.1 will launch the mesh generation phase of the analysis.

3. Next, the finite element analysis of the problem is run to compute sectional properties and stress distributions for given sectional loads. Clicking the second icon of the menu shown in fig. 1.1 will launch the finite element analysis of the section.

4. The last step of the process is the visualization of all results. Principal axes of bending, shearing and inertia are displayed together with the centroid, shear center and center of mass. Axial and shear stress or strain distributions associated with a given sectional loading can also be visualized. Clicking the third icon of the menu will invoke the visualization program, a detailed discussion of this topic appears in section 1.5.

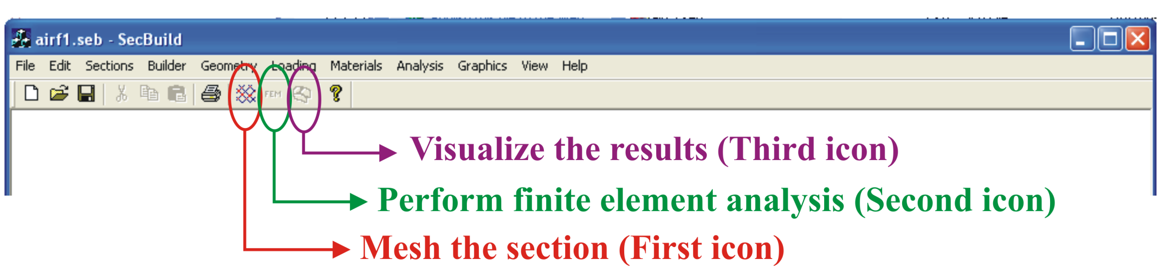

Figure 1.1: The SectionBuilder toolbar.

1.2 Definition of the beam cross-section

The definition of the configuration of the beam's cross-section involves two main components: the two-dimensional geometric configuration of the section, and the physical properties of the materials of which it is made.

- The geometric configuration of the cross-section can be defined in two alternative manners

  1. First, *parametric shapes* can be used, as discussed in detail in chapter 2. The following parametric shapes can be defined: airfoil sections as described in section 2.1, circular arcs as described in section 2.2, C-sections as described in section 2.3, circular cylinders as described in section 2.4, double boxes as described in section 2.5, I-sections as described in section 2.6, rectangular boxes as described in section 2.7, rectangular sections as described in section 2.8, circular tubes as described in section 2.9, triangular sections as described in section 2.10, and T-sections as described in section 2.11. The various configurations are parametrized, and hence, each section is readily defined by a small number of input parameters.
  2. Second, more *complex sections of arbitrary configuration* can be constructed, as discussed in chapter 3. The definition process is tailored towards the definition of composite structures: layers of material are stacked in a mold of arbitrary shape; ply insertions or drop-offs are allowed. The resulting unit is called a "wall." It is then possible to connect several walls to create complex sections of arbitrary configuration. The definition of walls is detailed in section 3.2; the following connectors are available: split-connectors as described in section 3.3, T-connectors as described in section 3.4 and V-connectors as described in section 3.5.

- The physical properties of the materials the section is made of can be defined in two alternative manners.

  1. Section 4.1 discusses the definition of *material properties.* Three types of materials can be defined: isotropic, orthotropic and transversely isotropic materials. Material stiffness, strength and density can be defined, and a failure criterion can be selected.
  2. Section 4.2 discusses the definition of *solid properties.* In this case, a layered material structure is defined; each layer features its proper material, ply thickness and fiber orientation angles.

1.3 Meshing the cross-section

Once the configuration of the cross-section has been defined a finite element mesh discretization is created by clicking the first icon of the SectionBuilder toolbar shown in fig. 1.1. A mapped mesh of the section is created. The meshing process recognizes the potential presence of layered materials: each layer is meshed independently to avoid smearing of the material properties. Meshes featuring finite elements of decreasing sizes can be created by specifying a mesh density parameter.

1.4 Performing the finite element analysis

Next, the mapped mesh generated in the previous step is used as the basis for a finite element analysis of the section launched by clicking the second icon of the SectionBuilder toolbar shown in fig. 1.1. The finite element analysis computes the three dimensional warping deformation field over the cross-section. Based on this warping field, the sectional stiffness and mass matrices are computed as well as three-dimensional stresses and strains at any location in the section. The predictions of the analysis are summarized in two files; the first details the sectional properties as described in section 1.4.1, and the second provides the the three dimensional stresses and strains at user specified location of the cross section as described in section 1.4.2.

1.4.1 Sectional properties

The geometry of the cross-section is described in an orthonormal basis $\mathcal{I} = (\bar{\imath}_1, \bar{\imath}_2, \bar{\imath}_3)$, where $\bar{\imath}_1$, $\bar{\imath}_2$ and $\bar{\imath}_3$ are three mutually orthogonal, unit vectors. The plane of the cross section is assumed to coincide with plane $(\bar{\imath}_2, \bar{\imath}_3)$, and the axis of the beam is along unit vector $\bar{\imath}_1$, as depicted in fig. 1.2. The reference axis of the beam is a line along axis $\bar{\imath}_1$; the origin of the coordinate system is at the intersection of the reference axis with the plane of the cross-section. This axis origin is not necessarily at the centroid and can be anywhere that is convenient. A detailed description of the sectional properties is printed in an output file of extension `.sbp` as described in section 1.6.3. This file contains the information detailed in the sections below.

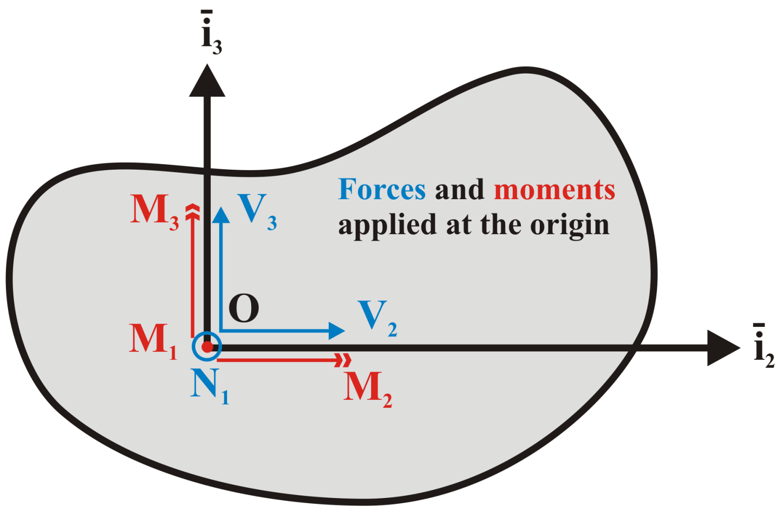

Figure 1.2: Sign conventions for the externally applied force and moment components acting on the cross-section.

Sectional stiffness and compliance matrices

The following sectional stiffness and compliance matrices are computed.

- The 4×4 *sectional stiffness matrix*. This matrix relates the sectional axial strain, ε_1, twisting curvature, κ_1, and two bending curvatures, κ_2 and κ_3, to the axial force, N_1, twisting moment, M_1, and two bending moments, M_2 and M_3. The relationship between these sectional strains and sectional stress resultants takes the form of a 4×4 matrix,

$$\left| \begin{matrix} N_1 \\ M_1 \\ M_2 \\ M_3 \end{matrix} \right| = \begin{bmatrix} C_{11} & C_{14} & C_{15} & C_{16} \\ C_{41} & C_{44} & C_{45} & C_{46} \\ C_{51} & C_{54} & C_{55} & C_{56} \\ C_{61} & C_{64} & C_{65} & C_{66} \end{bmatrix} \left| \begin{matrix} \varepsilon_1 \\ \kappa_1 \\ \kappa_2 \\ \kappa_3 \end{matrix} \right|. \tag{1.1}$$

- The 6×6 *sectional stiffness matrix*, C. This matrix relates the sectional axial strain, ε_1, transverse shearing strains, ε_2 and ε_3, twisting curvatures, κ_1 and two bending curvatures, κ_2 and κ_3, to the axial force, N_1, transverse shear forces, V_2 and V_3, twisting moment, M_1, and two bending moments, M_2 and M_3. The relationship between these sectional strains and sectional stress resultants takes the form of a symmetric, 6×6 matrix

$$\left| \begin{matrix} N_1 \\ V_2 \\ V_3 \\ M_1 \\ M_2 \\ M_3 \end{matrix} \right| = \begin{bmatrix} C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\ C_{12} & C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\ C_{13} & C_{23} & C_{33} & C_{34} & C_{35} & C_{36} \\ C_{14} & C_{24} & C_{34} & C_{44} & C_{45} & C_{46} \\ C_{15} & C_{25} & C_{35} & C_{45} & C_{55} & C_{56} \\ C_{16} & C_{26} & C_{36} & C_{46} & C_{56} & C_{66} \end{bmatrix} \left| \begin{matrix} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_3 \\ \kappa_1 \\ \kappa_2 \\ \kappa_3 \end{matrix} \right|, \text{ or } \underline{F} = C\, \underline{\epsilon}. \tag{1.2}$$

The three forces N_1, V_2 and V_3 are *positive along axes* $\bar{\imath}_1$, $\bar{\imath}_2$ and $\bar{\imath}_3$, respectively, whereas moments M_1, M_2 and M_3 are *positive about axes* $\bar{\imath}_1$, $\bar{\imath}_2$ and $\bar{\imath}_3$, respectively, as depicted in fig. 1.2. Identical sign conventions are used for the three strains, ε_1, ε_2 and ε_3, and curvatures κ_1, κ_2 and κ_3, respectively. The three forces are the resultants of the stress distributions over the cross section; the three moments are computed with respect to the origin of the coordinate system, *i.e.* with respect *to the reference axis of the beam*, as depicted in fig. 1.2.

- The 6×6 *sectional compliance matrix*, S, the inverse of the sectional stiffness matrix, *i.e.* $S = C^{-1}$, and hence,

$$\underline{\epsilon} = S\ \underline{F}. \tag{1.3}$$

It is often the case that the 6×6 stiffness matrix defined by eq. (1.2) contains a number of vanishing terms and presents the following structure

$$\left|\begin{array}{c} N_1 \\ V_2 \\ V_3 \\ M_1 \\ M_2 \\ M_3 \end{array}\right| = \left[\begin{array}{cccccc} C_{11} & 0 & 0 & 0 & C_{15} & C_{16} \\ 0 & C_{22} & C_{23} & C_{24} & 0 & 0 \\ 0 & C_{23} & C_{33} & C_{34} & 0 & 0 \\ 0 & C_{24} & C_{34} & C_{44} & 0 & 0 \\ C_{15} & 0 & 0 & 0 & C_{55} & C_{56} \\ C_{16} & 0 & 0 & 0 & C_{56} & C_{66} \end{array}\right] \left|\begin{array}{c} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_3 \\ \kappa_1 \\ \kappa_2 \\ \kappa_3 \end{array}\right|. \tag{1.4}$$

In such cases, the complete problem splits into an *axial force-bending moment* problem characterized by the following 3×3 stiffness matrix

$$\left|\begin{array}{c} N_1 \\ M_2 \\ M_3 \end{array}\right| = \left[\begin{array}{ccc} C_{11} & C_{15} & C_{16} \\ C_{15} & C_{55} & C_{56} \\ C_{16} & C_{56} & C_{66} \end{array}\right] \left|\begin{array}{c} \varepsilon_1 \\ \kappa_2 \\ \kappa_3 \end{array}\right|, \tag{1.5}$$

and a *twisting moment-shear force* problem characterized by the following 3×3 stiffness matrix

$$\left|\begin{array}{c} M_1 \\ V_2 \\ V_3 \end{array}\right| = \left[\begin{array}{ccc} C_{44} & C_{24} & C_{34} \\ C_{24} & C_{22} & C_{23} \\ C_{34} & C_{23} & C_{33} \end{array}\right] \left|\begin{array}{c} \kappa_1 \\ \varepsilon_2 \\ \varepsilon_3 \end{array}\right|. \tag{1.6}$$

The corresponding compliance matrices are

$$\left|\begin{array}{c} \varepsilon_1 \\ \kappa_2 \\ \kappa_3 \end{array}\right| = \left[\begin{array}{ccc} S_{11} & S_{15} & S_{16} \\ S_{15} & S_{55} & S_{56} \\ S_{16} & S_{56} & S_{66} \end{array}\right] \left|\begin{array}{c} N_1 \\ M_2 \\ M_3 \end{array}\right|, \tag{1.7}$$

and

$$\left|\begin{array}{c} \kappa_1 \\ \varepsilon_2 \\ \varepsilon_3 \end{array}\right| = \left[\begin{array}{ccc} S_{44} & S_{24} & S_{34} \\ S_{24} & S_{22} & S_{23} \\ S_{34} & S_{23} & S_{33} \end{array}\right] \left|\begin{array}{c} M_1 \\ V_2 \\ V_3 \end{array}\right|. \tag{1.8}$$

for the axial force-bending moment and twisting moment-shear force problems, respectively.

The axial force-bending moment problem

If the stiffness matrix of the cross-section presents the special structure displayed in eq. (1.4), it becomes possible to separately analyze the axial force-bending moment and twisting moment-shear force problems. The former problem is the focus of this section.

To further simplify the relationship between the axial forces and bending moment and the corresponding sectional strain components, eq. (1.5), it is convenient to introduce **the centroid of the cross-section**, a point of the cross-section with coordinates (x_{c2}, x_{c3}), as depicted in fig. 1.3. With the help of the centroid, the relationship between axial force and axial strain decouples from the relationship between bending moments and curvatures,

$$N_1^c = S\ \varepsilon_1^c. \tag{1.9}$$

where $N_1^c = N_1$ is the axial force, ε_1^c the axial strain at the centroid and S the axial stiffness. The bending moments are related to the sectional curvatures,

$$\left| \begin{array}{c} M_2^c \\ M_3^c \end{array} \right| = \left[\begin{array}{cc} H_{22}^c & -H_{23}^c \\ -H_{23}^c & H_{33}^c \end{array} \right] \left| \begin{array}{c} \kappa_2^c \\ \kappa_3^c \end{array} \right| \tag{1.10}$$

where M_2^c and M_3^c are the bending moments computed with respect to the centroid about axes parallel to $\bar{\imath}_2$ and $\bar{\imath}_3$, respectively, $\kappa_2^c = \kappa_2$ and $\kappa_3^c = \kappa_3$ the sectional curvatures, H_{22}^c and H_{33}^c the bending stiffnesses computed with respect to the centroid about axes parallel to $\bar{\imath}_2$ and $\bar{\imath}_3$, respectively, and H_{23}^c the cross bending stiffness computed with respect to the centroid about axes parallel to $\bar{\imath}_2$ and $\bar{\imath}_3$.

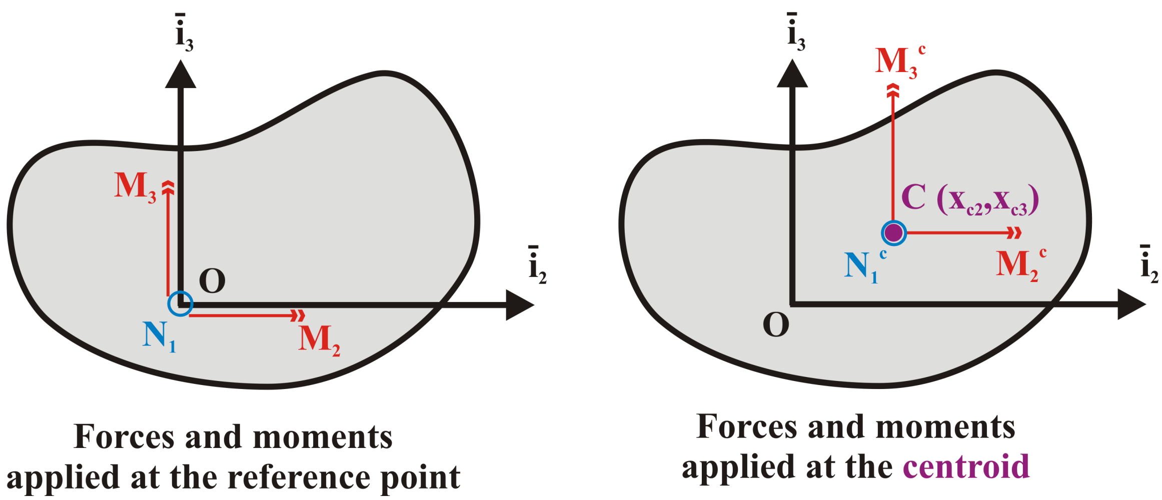

Figure 1.3: Left figure: forces and moments applied at the reference point. Right figure: forces and moments applied at the centroid.

The forces and moments computed with respect to the reference point and the centroid can be related as follows

$$\left| \begin{array}{c} N_1 \\ M_2 \\ M_3 \end{array} \right| = \left[\begin{array}{ccc} 1 & 0 & 0 \\ x_{c3} & 1 & 0 \\ -x_{c2} & 0 & 1 \end{array} \right] \left| \begin{array}{c} N_1^c \\ M_2^c \\ M_3^c \end{array} \right|; \qquad \left| \begin{array}{c} N_1^c \\ M_2^c \\ M_3^c \end{array} \right| = \left[\begin{array}{ccc} 1 & 0 & 0 \\ -x_{c3} & 1 & 0 \\ x_{c2} & 0 & 1 \end{array} \right] \left| \begin{array}{c} N_1 \\ M_2 \\ M_3 \end{array} \right|. \tag{1.11}$$

Similarly, the sectional strains and curvatures with respect to the reference point and the centroid can be related as follows

$$\left| \begin{array}{c} \varepsilon_1 \\ \kappa_2 \\ \kappa_3 \end{array} \right| = \left[\begin{array}{ccc} 1 & x_{c3} & -x_{c2} \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{array} \right] \left| \begin{array}{c} \varepsilon_1^c \\ \kappa_2^c \\ \kappa_3^c \end{array} \right|; \qquad \left| \begin{array}{c} \varepsilon_1^c \\ \kappa_2^c \\ \kappa_3^c \end{array} \right| = \left[\begin{array}{ccc} 1 & -x_{c3} & x_{c2} \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{array} \right] \left| \begin{array}{c} \varepsilon_1 \\ \kappa_2 \\ \kappa_3 \end{array} \right|. \tag{1.12}$$

Eqs. (1.9) and (1.10) relating the sectional forces and strains about the centroid can be recast in a single matrix equation as

$$\left| \begin{array}{c} N_1^c \\ M_2^c \\ M_3^c \end{array} \right| = \left[\begin{array}{ccc} S & 0 & 0 \\ 0 & H_{22}^c & -H_{23}^c \\ 0 & -H_{23}^c & H_{33}^c \end{array} \right] \left| \begin{array}{c} \varepsilon_1^c \\ \kappa_2^c \\ \kappa_3^c \end{array} \right|. \tag{1.13}$$

Introducing eqs. (1.11) and (1.12), the relationship between the corresponding quantities at the reference point are readily found as

$$\left| \begin{array}{c} N_1 \\ M_2 \\ M_3 \end{array} \right| = \left[\begin{array}{ccc} S & x_{c3}S & -x_{c2}S \\ x_{c3}S & H_{22}^c + x_{c3}^2 S & -(H_{23}^c + x_{c2}x_{c3}S) \\ -x_{c2}S & -(H_{23}^c + x_{c2}x_{c3}S) & H_{33}^c + x_{c2}^2 S \end{array} \right] \left| \begin{array}{c} \varepsilon_1 \\ \kappa_2 \\ \kappa_3 \end{array} \right|. \tag{1.14}$$

The corresponding bending compliance matrix is then found by inversion

$$\left|\begin{array}{c}\varepsilon_1\\ \kappa_2\\ \kappa_3\end{array}\right| = \frac{1}{\Delta_H}\left[\begin{array}{ccc}\frac{\Delta_H}{S}+x_{c2}^2H_{22}^c+x_{c3}^2H_{33}^c-2x_{c2}x_{c3}H_{23}^c & x_{c2}H_{23}^c-x_{c3}H_{33}^c & x_{c2}H_{22}^c-x_{c3}H_{23}^c\\ x_{c2}H_{23}^c-x_{c3}H_{33}^c & H_{33}^c & H_{23}^c\\ x_{c2}H_{22}^c-x_{c3}H_{23}^c & H_{23}^c & H_{22}^c\end{array}\right]\left|\begin{array}{c}N_1\\ M_2\\ M_3\end{array}\right|, \tag{1.15}$$

where $\Delta_H = H_{22}^cH_{33}^c - (H_{23}^c)^2$. Identifying the compliance matrices in eqs. (1.7) and (1.15), it is possible to compute from the terms of the compliance matrix the various engineering sectional stiffnesses listed below.

- The *centroidal bending stiffnesses*,

$$H_{22}^c = S_{66}/\Delta_S, \quad H_{33}^c = S_{55}/\Delta_S, \quad H_{23}^c = S_{56}/\Delta_S, \tag{1.16}$$

  where $\Delta_S = S_{55}S_{66} - S_{56}^2$.

- The coordinates of the *centroid location*,

$$x_{2c} = H_{33}^cS_{16} - H_{23}^cS_{15}, \quad x_{3c} = H_{23}^cS_{16} - H_{22}^cS_{15}. \tag{1.17}$$

- The *axial stiffness*,

$$S = \frac{1}{S_{11} - x_{2c}S_{16} + x_{3c}S_{15}}. \tag{1.18}$$

Eq. (1.10) describes the bending behavior of the cross-section, but due to the presence of the cross bending stiffness term, H_{23}^c, bending in the two planes $(\bar{\imath}_1, \bar{\imath}_2)$ and $(\bar{\imath}_1, \bar{\imath}_3)$ is coupled. It is possible to define the *principal centroidal axes of bending*. As illustrated in fig. 1.4, the principal centroidal axes of bending, $\mathcal{I}^{b*} = (\bar{\imath}_1^{b*}, \bar{\imath}_2^{b*}, \bar{\imath}_3^{b*})$, correspond to a planar rotation of orthonormal basis $\mathcal{I}$ by an angle α_b^*; note that clearly, $\bar{\imath}_1^{b*} = \bar{\imath}_1$. When using the principal centroidal axes of bending, the axial forces and bending moments are fully uncoupled,

$$N_1^{c*} = S\ \varepsilon_1^{c*}, \quad M_2^{c*} = H_{22}^{c*}\ \kappa_2^{c*}; \quad M_3^{c*} = H_{33}^{c*}\ \kappa_3^{c*}. \tag{1.19}$$

Clearly, $N_1^{c*} = N_1^c$ and $\varepsilon_1^{c*} = \varepsilon_1^c$, since the rotation of the axis system takes place about unit vector $\bar{\imath}_1$. The bending moments M_2^{c*} and M_3^{c*} are computed with respect to the centroid along axes $\bar{\imath}_2^{b*}$ and $\bar{\imath}_3^{b*}$, respectively. Similarly, κ_2^{c*} and κ_3^{c*} are the sectional curvatures about axes $\bar{\imath}_2^{b*}$ and $\bar{\imath}_3^{b*}$, respectively.

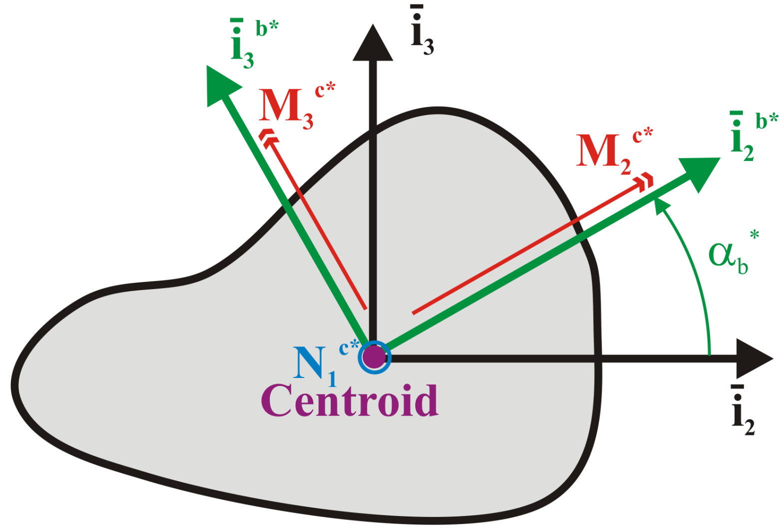

Figure 1.4: Orientation of the principal axes of bending.

The following quantities are also provided.

- The orientation, α_b^*, of the *principal centroidal axes of bending*,

$$\sin 2\alpha_b^* = \frac{H_{23}^c}{\Delta}, \quad \cos 2\alpha_b^* = \frac{H_{33}^c - H_{22}^c}{2\Delta}, \tag{1.20}$$

  where

$$\Delta = \sqrt{\left(\frac{H_{33}^c - H_{22}^c}{2}\right)^2 + (H_{23}^c)^2}. \tag{1.21}$$

- The *principal centroidal bending stiffnesses*, H_{22}^{c*} and H_{33}^{c*},

$$H_{22}^{c*} = \frac{H_{33}^c + H_{22}^c}{2} - \Delta, \quad H_{33}^{c*} = \frac{H_{33}^c + H_{22}^c}{2} + \Delta. \tag{1.22}$$

  Note that the choice the orientation of the principal axis $\bar{\imath}_2^{b*}$ given by eq. (1.20) guarantees that axis $\bar{\imath}_2^{b*}$ is the axis about which the minimum bending stiffness occurs; hence, $H_{22}^{c*} \leq H_{33}^{c*}$.

The twisting moment-shear force problem

If the stiffness matrix of the cross-section presents the special structure displayed in eq. (1.4), it becomes possible to separately analyze the axial force-bending moment and twisting moment shear force problems. The latter problem is the focus of this section.

To further simplify the relationship between the twisting moment and shearing forces and the corresponding sectional strain components, eq. (1.6), it is convenient to introduce **the shear center of the cross-section**, a point of the cross-section of coordinates (x_{k2}, x_{k3}), as depicted in fig. 1.5. With the help of the shear center, the relationship between twisting moment and twist rate decouples from the relationship between shearing forces and sectional transverse strains,

$$M_1^k = H_{11}^k \ \kappa_1^k; \tag{1.23}$$

where M_1^k is the twisting moment computed with respect to the shear center, $\varepsilon_1^k = \varepsilon_1$ the sectional twist rate and H_{11}^k the torsional stiffness. The shear forces are related to the sectional transverse strains,

$$\left|\begin{matrix} V_2^k \\ V_3^k \end{matrix}\right| = \left[\begin{matrix} K_{22}^k & -K_{23}^k \\ -K_{23}^k & K_{33}^k \end{matrix}\right] \left|\begin{matrix} \gamma_{12}^k \\ \gamma_{13}^k \end{matrix}\right| \tag{1.24}$$

where $V_2^k = V_2$ and $V_3^k = V_3$ are the sectional shearing forces, γ_{12}^k and γ_{13}^k the sectional transverse shearing strains, K_{22}^k and K_{33}^k the shearing stiffnesses computed with respect to the shear center about axes parallel to $\bar{\imath}_2$ and $\bar{\imath}_3$, respectively, and K_{23}^k the cross shearing stiffness computed with respect to the shear center about axes parallel to $\bar{\imath}_2$ and $\bar{\imath}_3$.

The forces and moments computed with respect to the reference point and the shear center can be related as follows

$$\left|\begin{matrix} M_1 \\ V_2 \\ V_3 \end{matrix}\right| = \left[\begin{matrix} 1 & -x_{k3} & x_{k2} \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{matrix}\right] \left|\begin{matrix} M_1^k \\ V_2^k \\ V_3^k \end{matrix}\right|; \qquad \left|\begin{matrix} M_1^k \\ V_2^k \\ V_3^k \end{matrix}\right| = \left[\begin{matrix} 1 & x_{k3} & -x_{k2} \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{matrix}\right] \left|\begin{matrix} M_1 \\ V_2 \\ V_3 \end{matrix}\right|. \tag{1.25}$$

Similarly, the sectional twist rate and transverse strains with respect to the reference point and shear center are related as follows

$$\left|\begin{matrix} \kappa_1 \\ \gamma_{12} \\ \gamma_{13} \end{matrix}\right| = \left[\begin{matrix} 1 & 0 & 0 \\ x_{k3} & 1 & 0 \\ -x_{k2} & 0 & 1 \end{matrix}\right] \left|\begin{matrix} \kappa_1^k \\ \gamma_{12}^k \\ \gamma_{13}^k \end{matrix}\right|; \qquad \left|\begin{matrix} \kappa_1^k \\ \gamma_{12}^k \\ \gamma_{13}^k \end{matrix}\right| = \left[\begin{matrix} 1 & 0 & 0 \\ -x_{k3} & 1 & 0 \\ x_{k2} & 0 & 1 \end{matrix}\right] \left|\begin{matrix} \kappa_1 \\ \gamma_{12} \\ \gamma_{13} \end{matrix}\right|. \tag{1.26}$$

Eqs. (1.23) and (1.24) relating the sectional forces and strains about the shear center can be recast in a matrix form as

$$\left|\begin{matrix} M_1^k \\ V_2^k \\ V_3^k \end{matrix}\right| = \left[\begin{matrix} H_{11}^k & 0 & 0 \\ 0 & K_{22}^k & -K_{23}^k \\ 0 & -K_{23}^k & K_{33}^k \end{matrix}\right] \left|\begin{matrix} \kappa_1^k \\ \gamma_{12}^k \\ \gamma_{13}^k \end{matrix}\right|. \tag{1.27}$$

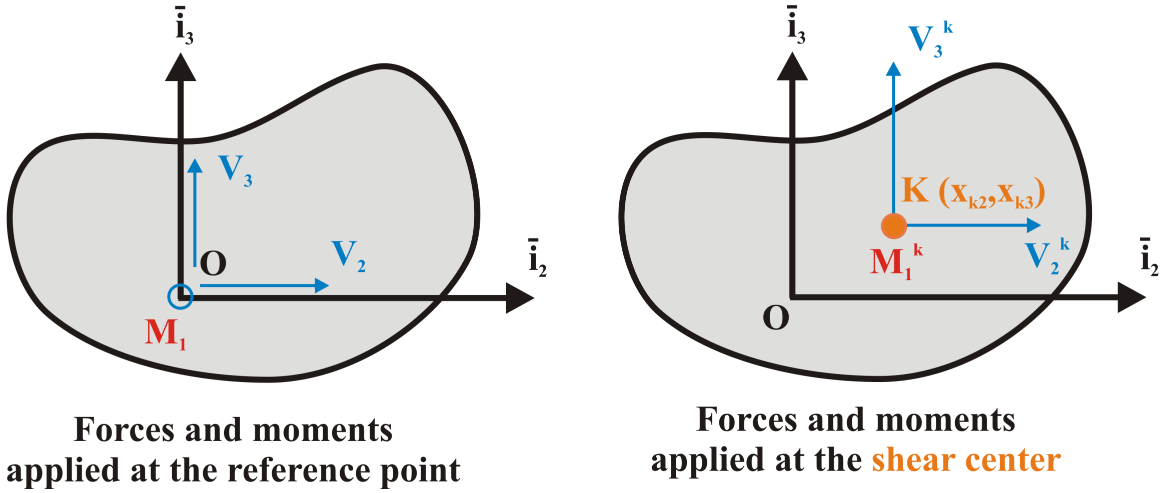

Figure 1.5: Left figure: forces and moments applied at the reference point. Right figure: forces and moments applied at the shear center.

Introducing eqs. (1.25) and (1.26), the relationship between the corresponding quantities at the reference point are readily found as

$$\left|\begin{array}{c} M_1 \\ V_2 \\ V_3 \end{array}\right| = \left[\begin{array}{ccc} H_{11} + x_{k2}^2 K_{33}^k + x_{k3}^2 K_{22}^k + 2x_{k2}x_{k3}K_{23}^k & -x_{k2}K_{23}^k - x_{k3}K_{22}^k & x_{k2}K_{33}^k + x_{k3}K_{23}^k \\ -x_{k2}K_{23}^k - x_{k3}K_{22}^k & K_{22}^k & -K_{23}^k \\ x_{k2}K_{33}^k + x_{k3}K_{23}^k & -K_{23}^k & K_{33}^k \end{array}\right] \left|\begin{array}{c} \kappa_1 \\ \gamma_{12} \\ \gamma_{13} \end{array}\right| \tag{1.28}$$

The corresponding bending compliance matrix is then found by inversion

$$\left|\begin{array}{c} \kappa_1 \\ \gamma_{12} \\ \gamma_{13} \end{array}\right| = \left[\begin{array}{ccc} 1/H_{11} & x_{k3}/H_{11} & -x_{k2}/H_{11} \\ x_{k3}/H_{11} & K_{33}^k/\Delta_K + x_{k3}^2/H_{11} & K_{23}^k/\Delta_K - x_{k2}x_{k3}/H_{11} \\ -x_{k2}/H_{11} & K_{23}^k/\Delta_K - x_{k2}x_{k3}/H_{11} & K_{22}^k/\Delta_K + x_{k2}^2/H_{11} \end{array}\right] \left|\begin{array}{c} M_1 \\ V_2 \\ V_3 \end{array}\right|, \tag{1.29}$$

where $\Delta_K = K_{22}^k K_{33}^k - (K_{23}^k)^2$. Identifying the compliance matrices in eqs. (1.8) and (1.29), it is possible to compute from the terms of the compliance matrix the various engineering sectional stiffnesses listed below.

- The *torsional stiffness*,

$$H_{11} = \frac{1}{S_{44}}. \tag{1.30}$$

- The coordinates of the *shear center*,

$$x_{2k} = -H_{11}S_{34}, \quad x_{3k} = H_{11}S_{24}. \tag{1.31}$$

- The *shearing stiffnesses about shear center*,

$$K_{22}^k = b\Delta_K, \quad K_{33}^k = a\Delta_K, \quad K_{23}^k = c\Delta_K, \tag{1.32}$$

  where $a = S_{22} - x_{3k}^2/H_{11}$, $b = S_{33} - x_{2k}^2/H_{11}$, $c = S_{23} + x_{2k}x_{3k}/H_{11}$ and $\Delta = 1/(ab - c^2)$.

Eq. (1.24) describes the shearing behavior of the cross-section, but due to the presence of the cross shearing stiffness term, K_{23}^k, shearing in the two planes $(\bar{\imath}_1, \bar{\imath}_2)$ and $(\bar{\imath}_1, \bar{\imath}_3)$ is coupled. It is possible to define the *principal axes of shearing at the shear center*. As illustrated in fig. 1.6, the principal axes of shearing at the shear center, $\mathcal{I}^* = (\bar{\imath}_1^{s*}, \bar{\imath}_2^{s*}, \bar{\imath}_3^{s*})$, correspond to a planar rotation of orthonormal basis $\mathcal{I}$ by an angle α_s^*; note that clearly, $\bar{\imath}_1^{s*} = \bar{\imath}_1$. When using the principal axes of shearing at the shear center, the twisting moment and shearing forces are fully uncoupled,

$$M_1^{k*} = H_{11}^k\ \kappa_1^{k*}, \quad V_2^{k*} = K_{22}^{k*}\ \gamma_{12}^{k*}; \quad V_3^{k*} = K_{33}^{k*}\ \gamma_{13}^{k*}. \tag{1.33}$$

Clearly, $M_1^{k*} = M_1^k$ and $\kappa_1^{k*} = \kappa_1^k$, since the rotation of the axis system takes place about unit vector $\bar{\imath}_1$. The shearing forces V_2^{k*} and V_3^{k*} are computed with respect to the shear center along axes $\bar{\imath}_2^{s*}$ and $\bar{\imath}_3^{s*}$, respectively. Similarly, γ_{12}^{k*} and γ_{13}^{k*} are the sectional transverse strains along axes $\bar{\imath}_2^{s*}$ and $\bar{\imath}_3^{s*}$, respectively.

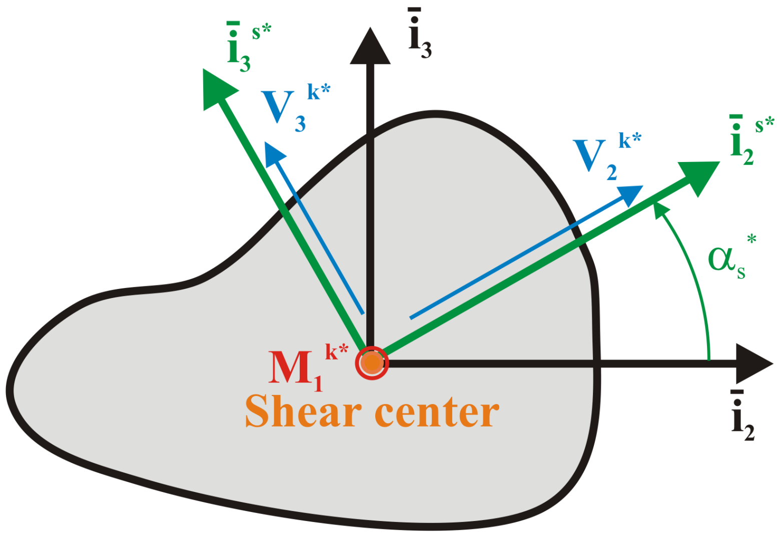

Figure 1.6: Orientation of the principal axes of shearing.

The following quantities are also provided.

- The orientation, α_s^*, of the *principal axes of shearing at the shear center*,

$$\sin 2\alpha_s^* = \frac{K_{23}^k}{\Delta}, \quad \cos 2\alpha_s^* = -\frac{K_{33}^k - K_{22}^k}{2\Delta}, \tag{1.34}$$

  where

$$\Delta = \sqrt{\left(\frac{K_{33}^k - K_{22}^k}{2}\right)^2 + (K_{23}^k)^2}. \tag{1.35}$$

- The *principal shearing stiffnesses at the shear center*, K_{22}^{k*} and K_{33}^{k*},

$$K_{22}^{k*} = \frac{K_{33}^k + K_{22}^k}{2} - \Delta, \quad K_{33}^{k*} = \frac{K_{33}^k + K_{22}^k}{2} + \Delta. \tag{1.36}$$

  Note that the choice the orientation of the principal axis $\bar{\imath}_2^{s*}$ given by eq. (1.34) guarantees that axis $\bar{\imath}_2^{s*}$ is the axis along which the minimum shearing stiffness occurs; hence, $K_{22}^{k*} \leq K_{33}^{k*}$.

Sectional masses and moments of inertia

The 6×6 *sectional mass matrix*, M. This matrix relates the sectional linear velocities, denoted v_1, v_2 and v_3, and angular velocities, denoted ω_1, ω_2 and ω_3, to the sectional linear momenta, denoted p_1, p_2 and p_3, and angular momenta, denoted h_1, h_2 and h_3. The relationship between these sectional velocities and sectional momenta takes the form of a symmetric, 6×6 matrix

$$\left\lvert \begin{array}{c} p_1 \\ p_2 \\ p_3 \\ h_1 \\ h_2 \\ h_3 \end{array} \right\rvert = \begin{bmatrix} M_{11} & M_{12} & M_{13} & M_{14} & M_{15} & M_{16} \\ M_{12} & M_{22} & M_{23} & M_{24} & M_{25} & M_{26} \\ M_{13} & M_{23} & M_{33} & M_{34} & M_{35} & M_{36} \\ M_{14} & M_{24} & M_{34} & M_{44} & M_{45} & M_{46} \\ M_{15} & M_{25} & M_{35} & M_{45} & M_{55} & M_{56} \\ M_{16} & M_{26} & M_{36} & M_{46} & M_{56} & M_{66} \end{bmatrix} \left\lvert \begin{array}{c} v_1 \\ v_2 \\ v_3 \\ \omega_1 \\ \omega_2 \\ \omega_3 \end{array} \right\rvert . \tag{1.37}$$

Due to the nature of the problem, many of these coefficients vanish, and the remaining entries are written as

$$\begin{vmatrix} p_1 \\ p_2 \\ p_3 \\ h_1 \\ h_2 \\ h_3 \end{vmatrix} = \begin{bmatrix} m_{00} & 0 & 0 & 0 & m_{00}x_{3m} & -m_{00}x_{2m} \\ 0 & m_{00} & 0 & -m_{00}x_{3m} & 0 & 0 \\ 0 & 0 & m_{00} & m_{00}x_{2m} & 0 & 0 \\ 0 & -m_{00}x_{3m} & m_{00}x_{2m} & I_{11} & 0 & 0 \\ m_{00}x_{3m} & 0 & 0 & 0 & I_{22} & I_{23} \\ -m_{00}x_{2m} & 0 & 0 & 0 & I_{23} & I_{33} \end{bmatrix} \begin{vmatrix} v_1 \\ v_2 \\ v_3 \\ \omega_1 \\ \omega_2 \\ \omega_3 \end{vmatrix}, \quad (1.38)$$

where m_{00} is the sectional mass per unit span, x_{2m} and x_{3m} the coordinates of the center of mass, I_{22}, I_{33} and I_{23} the components of the sectional mass moments of inertia per unit span, and I_{11} the sectional polar moment of inertia per unit span.

The following quantities are also provided.

- The *sectional area*, $\mathcal{A}$.
- The *sectional mass per unit span*,

$$m_{00} = M_{11} = M_{22} = M_{33}. \quad (1.39)$$

- The location of the *mass center*,

$$x_{2m} = \frac{M_{34}}{m_{00}}, \quad x_{3m} = \frac{M_{15}}{m_{00}}. \quad (1.40)$$

- The *mass moments of inertia per unit span* about the center of mass,

$$I_{22}^m = I_{22} - m_{00}x_{m3}^2, \quad I_{33}^m = I_{33} - m_{00}x_{m2}^2, \quad I_{23}^m = I_{23} + m_{00}x_{m2}x_{m3}. \quad (1.41)$$

- The *polar moment of inertia per unit span* about the center of mass,

$$I_{11}^m = I_{22}^m + I_{33}^m. \quad (1.42)$$

It is possible to define the *principal axes of inertia at the center of mass.* As illustrated in fig. 1.7, the principal axes of inertia at the center of mass, $\mathcal{I}^{m*} = (\bar{\imath}_1^{m*}, \bar{\imath}_2^{m*}, \bar{\imath}_3^{m*})$, correspond to a planar rotation of orthonormal basis $\mathcal{I}$ by an angle α_m^*; about $\bar{\imath}_1$ so that, $\bar{\imath}_1^{m*} = \bar{\imath}_1$. When using the principal axes of inertia at the center of mass, the relationship between angular momenta and angular velocities so that uncouples

$$h_1^{m*} = I_{11}^m\ \omega_1^*, \quad h_2^{m*} = I_{22}^{m*}\ \omega_2^*; \quad h_3^{m*} = I_{33}^{m*}\ \omega_3^*. \quad (1.43)$$

Clearly, $h_1^{m*} = h_1^m$ and $\omega_1^* = \omega_1$, since the rotation of the axis system takes place about unit vector $\bar{\imath}_1$. The angular momenta h_2^{m*} and h_3^{m*} are computed with respect to the center of mass along axes $\bar{\imath}_2^{m*}$ and $\bar{\imath}_3^{m*}$, respectively. Similarly, ω_2^* and ω_3^* are the angular velocities about axes $\bar{\imath}_2^{m*}$ and $\bar{\imath}_3^{m*}$, respectively.

The following quantities are also provided.

- The orientation, α_m^*, of the *principal axes of inertia at the center of mass*,

$$\sin 2\alpha_m^* = -\frac{I_{23}^m}{\Delta}, \quad \cos 2\alpha_m^* = \frac{I_{33}^m - I_{22}^m}{2\Delta}, \quad (1.44)$$

where

$$\Delta = \sqrt{\left(\frac{I_{33}^m - I_{22}^m}{2}\right)^2 + (I_{23}^m)^2}. \quad (1.45)$$

- The *principal moments of inertia per unit span about center of mass*, I_{22}^{m*} and I_{33}^{m*},

$$I_{22}^{m*} = \frac{I_{33}^m + I_{22}^m}{2} - \Delta, \quad I_{33}^{m*} = \frac{I_{33}^m + I_{22}^m}{2} + \Delta. \quad (1.46)$$

Note that the choice the orientation of the principal axis $\bar{\imath}_2^{m*}$ given by eq. (1.44) guarantees that axis $\bar{\imath}_2^{m*}$ is the axis about which the minimum moments of inertia occurs; hence, $I_{22}^{m*} \leq I_{33}^{m*}$.

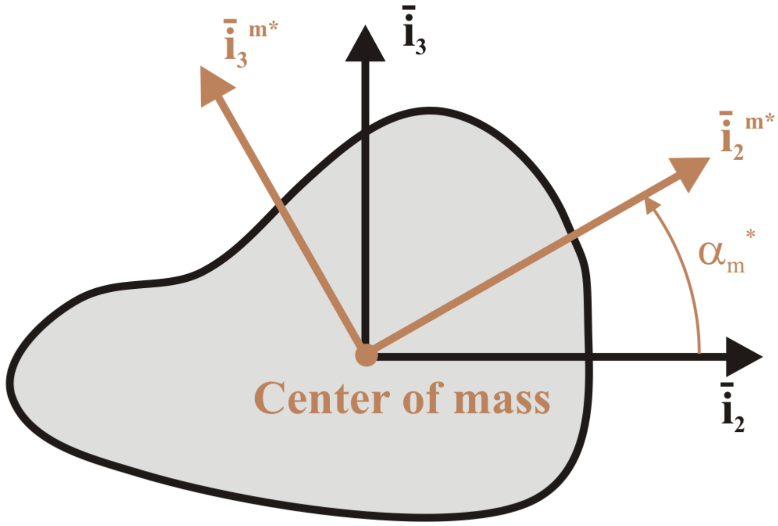

Figure 1.7: Orientation of the principal axes of inertia at the center of mass.

1.4.2 Three-dimensional stresses and strains

The geometry of the cross-section is described in an orthonormal basis $\mathcal{I} = (\bar{\imath}_1, \bar{\imath}_2, \bar{\imath}_3)$, where $\bar{\imath}_1$, $\bar{\imath}_2$ and $\bar{\imath}_3$ are three mutually orthogonal, unit vectors. The plane of the cross section is assumed to coincide with plane $(\bar{\imath}_2, \bar{\imath}_3)$, and the axis of the beam is along unit vector $\bar{\imath}_1$, as depicted in fig. 1.2. The reference axis of the beam is a line along axis $\bar{\imath}_1$; the origin of the coordinate system is at the intersection of the reference axis with the plane of the cross-section.

To compute three-dimensional warping displacement, stress components or strain components, two user defined inputs are required.

1. First, *sectional loading cases* must be defined as described in section 6.1. A loading case consists of an axial force and two transverse shear forces, as well as a twisting moment and two bending moments, applied to the cross-section. These 3 forces and 3 moments can be applied at the reference axis, at the centroid, at the shear center, or at an arbitrary point of the cross-section.

2. Second, *sensors* must be defined as described in section 8.5. These "sensors" are analogous to their physical counterparts, such as strain gauges, which provide information about the local strain field. Sensors define the location on the cross-section where the information will be computed and the type of quantity to be sensed, which could be three-dimensional warping displacement, stress components or strain components.

For each of the defined loading cases, the quantities measured by each of the sensors will be computed and printed. A detailed report is printed in an output file with extension `.sbs` as described in section 1.6.3.

Warping displacements

Under the effect of the applied loading, the cross-section will deform. This deformation is characterized by a three-dimensional warping displacement field, which features components both in- and out-of-plane of the cross-section. A typical print-out of a warping sensor is shown in Table 1.1. For the sensor named `SensorWarping`, the out-of-plane warping displacement component, w_1, as well as the in-plane warping displacement components, w_2 and w_3, are listed at the sensor location. The displacement components w_1, w_2 and w_3 are the components of the displacement vector along unit vectors $\bar{\imath}_1$, $\bar{\imath}_2$ and $\bar{\imath}_3$, respectively.

— Sensor : [SensorWarping].		
Sensor location (x_2, x_3)	=	(-1.9e-18, -7.4e-02) [m].
Section warping: w_1, w_2, w_3	=	(0.0e+00, -3.9e-09, 4.8e-09) [m].

Table 1.1: Print-out of the three-dimensional warping displacement at a sensor location.

Three-dimensional stresses

Under the effect of the applied loading, the cross-section will deform, generating a three-dimensional stress field, which features both in- and out-of-plane components on the cross-section. Since the cross-section is in plane $(\bar{\imath}_2, \bar{\imath}_3)$, the in-plane stress components are σ_2, τ_{23} and σ_3, whereas the out-of-plane stress components are σ_1, τ_{12} and τ_{13} as described in fig. 1.8. Note that classical beam theory predicts only the the out-of-plane stress component, σ_1, and the two transverse shearing stresses, τ_{12} and τ_{13}; typically, the in-plane stress components are assumed to be negligible, *i.e.* $\sigma_2 \approx 0$, $\tau_{23} \approx 0$ and $\sigma_3 \approx 0$. The analysis implemented in `SectionBuilder` predicts both out-of-plane and in-plane stress components.

A typical print-out of a stress sensor is shown in Table 1.2. For the sensor named `SensorStresses`, the out-of-plane stress components, σ_1, τ_{12} and τ_{13}, as well as the in-plane stress components, σ_2, τ_{23} and σ_3, are listed at the sensor location. Fig. 1.8 shows the sign convention used for the stress computation.

--- Sensor : [SensorStresses].		
Sensor location (x_2, x_3)	=	(2.7e-02, -4.7e-02) [m].
Out-of-plane stresses: σ_1, τ_{12}, τ_{13}	=	(3.0e+04, 0.0e+00, 0.0e+00) [Pa].
In-plane stresses: σ_2, τ_{23}, σ_3	=	(-1.7e-10, 9.9e-11, 1.2e-10) [Pa].
Reserve factors	=	(2.0e+04, -2.0e+04)

Table 1.2: Print-out of the three-dimensional strains at a sensor location.

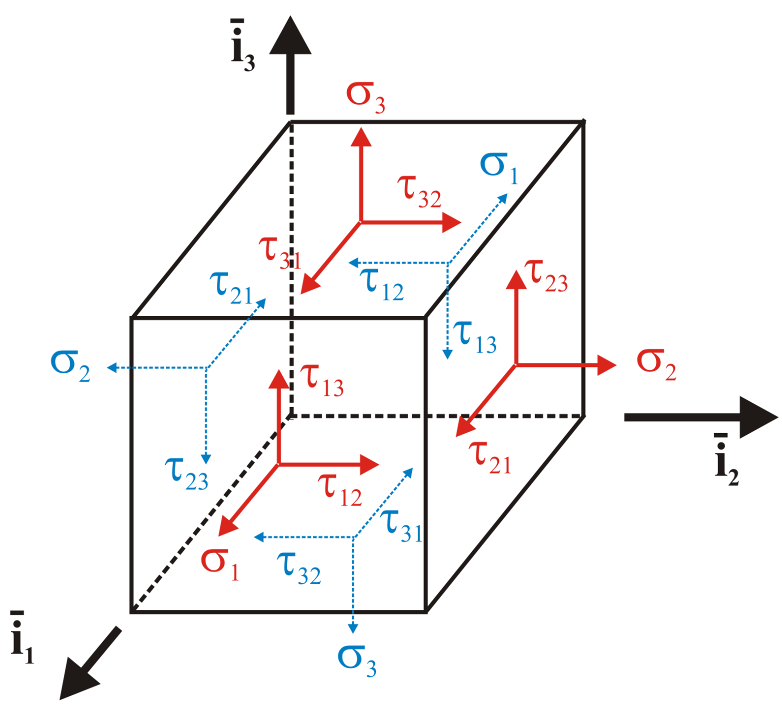

Figure 1.8: Sign convention for the three-dimensional stresses acting on a differential element of the beam.

Three-dimensional strains

Under the effect of the applied loading, the cross-section will deform, generating a three-dimensional strain field, which features both in- and out-of-plane components on the cross-section. Since the cross-section is in plane $(\bar{\imath}_2, \bar{\imath}_3)$, the in-plane strain components are ϵ_2, γ_{23} and ϵ_3, whereas the out-of-plane strain components are ϵ_1, γ_{12} and γ_{13}. Note that classical beam theory predicts only the the out-of-plane strain component, ϵ_1, and the two transverse shearing strains, γ_{12} and γ_{13}; typically, the in-plane strain components are ignored because the Euler-Bernoulli kinematic assumptions correspond to a rigid body motion of the cross-section, *i.e.* $\epsilon_2 = 0$, $\gamma_{23} = 0$ and $\epsilon_3 = 0$. The analysis implemented in `SectionBuilder` predicts both out-of-plane and in-plane strain components.

A typical print-out of a strain sensor is shown in Table 1.3. For the sensor named `SensorStrains`, the out-of-plane strain components, ϵ_1, γ_{12} and γ_{13}, as well as the in-plane strain components, ϵ_2, γ_{23} and ϵ_3, are listed at the sensor location. The sign conventions for strain components are consistant with the stresses shown for stresses in fig. 1.8.

`--- Sensor :  [SensorStrains].`		
`Sensor location` (x_2, x_3)	=	`( 2.7e-02, -4.7e-02) [m].`
`Out-of-plane strains:` ϵ_1, γ_{12}, γ_{13}	=	`( 4.2e-07, 0.0e+00, 0.0e+00)`
`In-plane strains:` ϵ_2, γ_{23}, ϵ_3	=	`(-1.2e-07, 3.5e-21, -1.2e-07)`
`Reserve factors`	=	`( 2.0e+04, -2.0e+04)`

Table 1.3: Print-out of the three-dimensional strains at a sensor location.

1.5 Visualizing the results

The last step of the SectionBuilder process is to visualize the results of the finite element analysis performed in the previous step. Clicking the third icon of the SectionBuilder toolbar shown in fig. 1.1 enters the visualization mode. Some of the results of the finite element analysis, such as the sectional stiffness and compliance matrices, do not lend themselves to visualization, however, many of the other computed quantities are most easily interpreted through graphic visualization.

Visualization of the results is controlled by two menu items and associated toolbars: the `Loading` toolbar, shown in fig. 1.9, and the `Graphics` toolbar, shown in fig. 1.12. Visualization proceeds in three steps controlled by the the `Loading` toolbar.

1. First, select a *sectional loading case* as described in section 6.1. This is an essential step because the warping, stress or strain fields all depend on the applied loading. More details are given in section 1.5.1.

2. Second, select the *quantities to be visualized*; they fall into two main groups: *(1)* sectional centers and principal axes and *(2)* the warping, stress or strain fields over the cross-section. More details are given in section 1.5.2.

3. Optionally, it is also possible to interactively *define sensors* as described in section 8.5, at specific locations over the cross-section, as discussed in section 1.5.3.

The `Graphics` toolbar controls the manner in which the requested information is to be visualized, and more details are given in section 1.5.4.

1.5.1 Step 1: Selecting a sectional loading case

The first step of the visualization phase is to select a sectional loading condition, as provided by the sectional loads(see section 6.1). These user defined loading conditions form a list of loading conditions. The first three entries or icons of the `Loading` menu shown in fig. 1.9 are used to navigate this loading list.

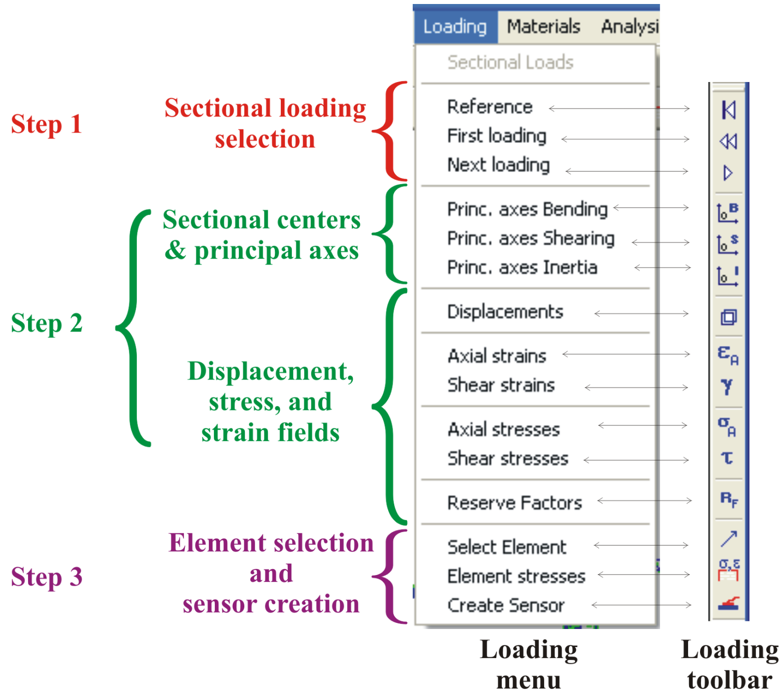

Figure 1.9: The *Loading* menu and toolbar. Each action can be invoked by selecting a menu item or clicking the corresponding toolbar icon. In this figure, the menu items and corresponding toolbar icons are shown next to each other to highlight the correspondence.

1. The `Reference` menu item or toolbar icon will show the reference configuration of the cross-section, no loading condition is selected.
2. The `First Loading` menu item or toolbar icon activates the first loading condition in the list.
3. The `Next Loading` menu item or toolbar icon moves to the next loading condition in the list.

1.5.2 Step 2: Selecting the quantities to be visualized

The second step of the visualization phase is to select the quantities to be visualized. Nine entries or icons in the `Loading` menu shown in fig. 1.9 are used to select the desired quantities to be visualized, and these fall into two categories: *(1)* sectional centers and principal axes and, *(2)* the warping, stress or strain fields over the cross-section.

Visualizing centers and principal axes

The first three entries or icons are used to visualize the centroid, shear center and center of mass of the cross-section and the associated principal axes. These entries or icons are toggle switches that turn on and off the visualization of the associated quantities. In all cases, the origin of the axis system, *i.e.* the reference axis of the beam is indicated by a yellow circle.

1. The `Princ. axes Bending` menu item or toolbar icon switches on the visualization of the *principal centroidal axes of bending*. The location of the centroid is given by eq. (1.17) and the orientation of the principal centroidal axes by eq. (1.20), as illustrated in fig. 1.4. Examples are shown in fig. 2.10 and fig. 2.65.
2. The `Princ. axes Shearing` menu item or toolbar icon switches on the visualization of the *principal axes of shearing at the shear center*. The location of the shear center is given by eq. (1.31) and the

orientation of the principal axes of shearing at the shear center by eq. (1.34), as illustrated in fig. 1.6. Examples are shown in fig. 2.16 and fig. 2.85.

3. The `Princ. axes Inertia` menu item or toolbar icon switches on the visualization of the *principal axes of inertia at the mass center*. The location of the mass center is given by eq. (1.40) and the orientation of the principal axes of inertia at the mass center by eq. (1.44), as illustrated in fig. 1.7. Examples are shown in fig. 2.11 and fig. 2.79.

As discussed in section 1.4.1, the complete sectional stress-sectional strain relationship given by eq. (1.2) often splits into an *axial force-bending moment* problem, characterized by eq. (1.5), and a *twisting moment-shear force* problem, characterized by eq. (1.6). When this decoupling takes place, the `Princ. axes Bending` and `Princ. axes Shearing` entries or icons will display the quantities indicated above. If the decoupling does not take place, clicking these entries or icons will simply display the origin of the axis system, since the corresponding centers and associated principal axes do not exist.

Visualizing warping, stress and strain fields

The next six entries or icons control the display of the warping displacement, stress and strain fields over the cross-section. Clicking one of these entries or icons will cause the display of a specific displacement, stress or strain component; the six actions are mutually exclusive.

1. Clicking the `Displacements` menu item or toolbar icon causes the display of the three-dimensional *warping displacement* of the cross-section under the applied sectional loads. Examples are shown in fig. 2.38 and fig. 2.78.

2. Clicking the `Axial strains` menu item or toolbar icon causes the display of the *axial strain field* over the cross-section under the applied sectional loads. Examples are shown in fig. 2.44 and fig. 2.56.

3. Clicking the `Shear strains` menu item or toolbar icon causes the display of the *shear strain field* over the cross-section under the applied sectional loads. Examples are shown in fig. 2.17 and fig. 2.29.

4. Clicking the `Axial stresses` menu item or toolbar icon causes the display of the *axial stress field* over the cross-section under the applied sectional loads. Examples are shown in fig. 2.30, fig. 2.45, fig. 2.66 and fig. 2.92.

5. Clicking the `Shear stresses` menu item or toolbar icon causes the display of the *shear stress field* over the cross-section under the applied sectional loads. Examples are shown in fig. 2.55 and fig. 2.73.

6. Clicking the `Reserve Factors` menu item or toolbar icon causes the display of the reserve factor field over the cross-section under the applied sectional loads. When the menu item or toolbar icon `Reserve Factors` is pressed, the reserve factors are computed over the cross-section, and the **inverse of the reserve factor** is displayed. Examples are shown in fig. 2.72 and fig. 2.91.

The visualization of the centers and principal axes can be superimposed onto the display of these various fields.

1.5.3 Step 3: Interactive definition of sensors

The last step of the visualization phase is the optional interactive definition of sensors. The last three entries or icons of the `Loading` menu shown in fig. 1.9 are used to that effect; two options are possible: *(1)* a temporary display the warping displacement, stress or strain levels at a point of the cross-section, or *(2)* the interactive definition of sensors.

Temporary display of warping displacement, stress or strain levels

It is often important to obtain numerical, rather than graphical information about the warping displacement, stress or strain levels at a point of the cross-section. This can be done by selecting a specific finite element from the model, then displaying the stresses or strain at that point. The procedure is as follows.

1. Enter the visualization mode, select a loading condition, and visualize warping displacement, stress or strain fields. These steps are described in sections 1.5.1 and 1.5.2.

2. To select an element of the finite element mesh, first click the `Select Element` menu item or toolbar icon, then click on the desired element; that element will be highlighted as shown in fig. 1.10.

3. Next, click the `Element Stresses` menu item or toolbar icon to display the stresses or strain at the center of the element. If the warping displacement, stress or strain field is presently visualized, warping displacement, stress or strain levels will be displayed, respectively.

Fig. 1.10 illustrate this process while the shear stress field is visualized. An element of the mesh is selected, and is highlighted, as shown on the figure. The temporary pop-up display gives the three-dimensional stress components at the center point of the selected element. Once the `OK` icon is clicked, the box disappears, and no record is kept of the stress level at that point. If a permanent record of the stress level is desired, the interactive definition of a sensor should be used, as discussed in the next section.

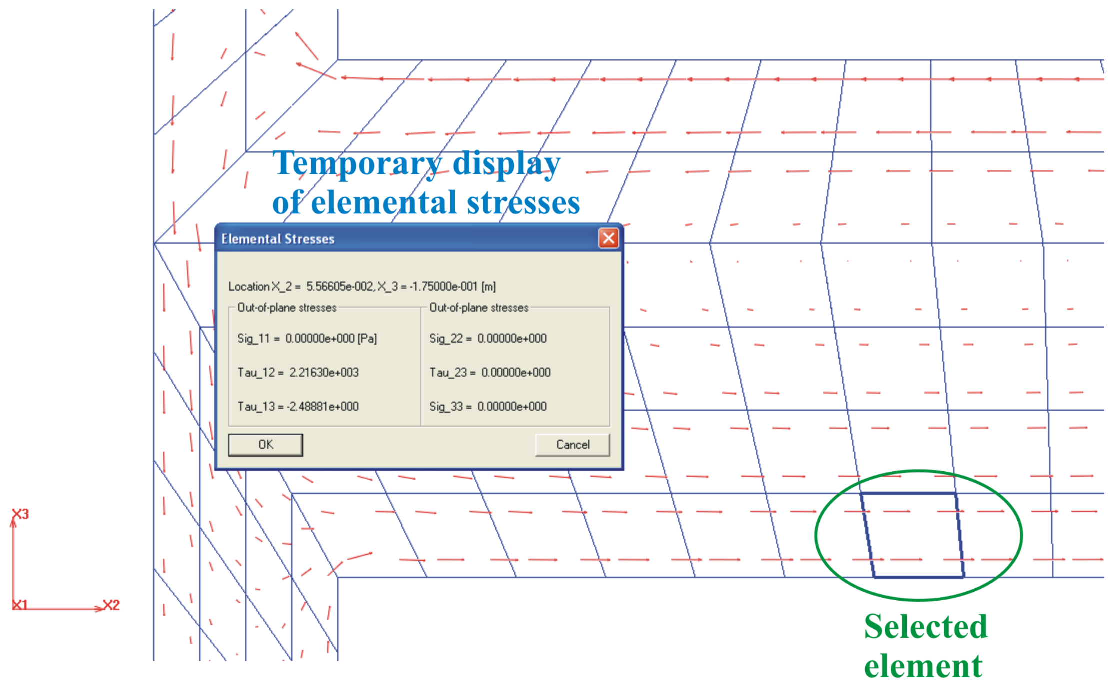

Figure 1.10: The temporary display of stress levels.

Interactive definition of sensors

It is often convenient to create a permanent record of the warping displacement, stress or strain levels at a point of the cross-section. This can be achieved through the interactive definition of sensors as described in section 8.5, at specific locations of the cross-section. Once defined, these sensors become part of the model definition and, as discussed in section 1.4.2, each time the finite element analysis is run, warping displacements, stresses or strains will be output for each sensor and each loading condition.

To define a sensor, first select the desired element of the model, then create a sensor at that location. The procedure is as follows.

1. Enter the visualization mode, select a loading condition, and visualize warping displacement, stress or strain fields. These steps are described in sections 1.5.1 and 1.5.2.

2. To select an element of the finite element mesh, first click the `Select Element` menu item or toolbar icon, then click on the desired element; that element will be highlighted as shown in fig.1.11.

3. Next, click the `Create Sensor` menu item or toolbar icon to open the interactive sensor definition dialog shown in fig.1.11 an fill in the following information.

   (a) **Sensor name.** Enter a unique name for the sensor.

   (b) **Sensor type.** Select *Stresses*, *Strains* or *Warping* for the sensor to compute the six stress components, the six strain components, or the three warping components, respectively.

   (c) **Sensor Location.** Select *Middle*, *At Gauss Points* or *At Corners* to locate one sensor at the middle of the element, four sensors at each of the four Gauss points of the element, or four sensors at the four corners of the element, respectively. For stresses and strains, the Gauss point option provides the most accurate information, whereas for warping displacements, the four corner option is preferable.

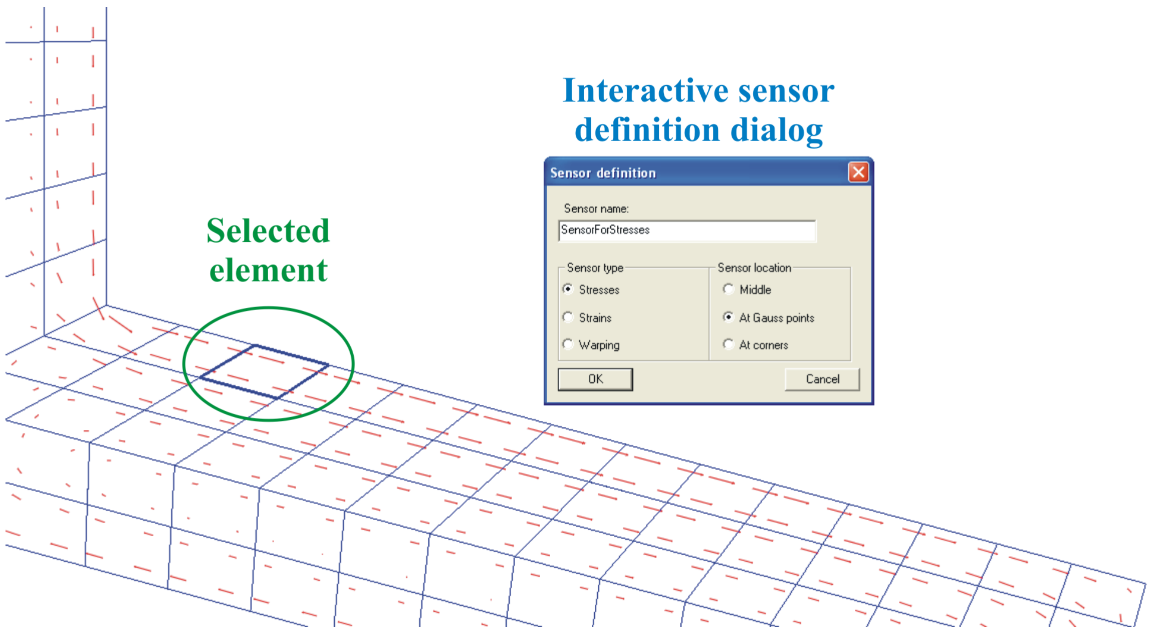

Figure 1.11: The interactive definition of a sensor.

1.5.4 The *Graphics* menu and toolbar

The *Graphics* menu and toolbar shown in fig. 1.12 controls the manner in which the requested information is to be visualized.

1. The `Zoom In` and `Zoom Out` perform the familiar zoom in and zoom out function to focus on a specific portion of the cross-section (zoom in), or obtain an overall view of the entire cross-section (zoom out).

2. The `Center` menu item or toolbar icon brings the geometric center of the cross-section to the center of the graphical window, whereas the `Show Model` menu item or toolbar icon adjusts the zooming level so the entire cross-section is visible in the graphical window.

3. Although a two-dimensional slice of beam is defined in the analysis, three-dimensional displacement, stress, and strain fields are computed. To allow visualization of stress and strain components, it is often necessary to rotate the model in the graphical window. The six entries or icons, `Rotate X+`, `Rotate X-`, `Rotate Y+`, `Rotate Y-`, `Rotate Z+` and `Rotate Z-` rotate the model by ± 5 degrees about the X, Y and Z axes, respectively. The X, Y and Z axes are **screen axes**.

4. Similarly, the four entries or icons, `Translate X+`, `Translate X-`, `Translate Y+` and `Translate Y-` translate the model by ±5% along the X or Y axes, respectively.

5. The `View From X1` menu item or toolbar icon reset the view point so that the structural axis $\bar{\imath}_1$ is now perpendicular to the plane of the screen.

6. The size of the symbols used in the visualization can be adjusted using the `Symbol size +` or `Symbol size -` entries or icons. The symbols are the circles used to indicate the locations of the sectional centers and the arrays that represent the principal axes.

7. The size of the arrows used to visualize the strain or stress fields can be adjusted using the `Data Size +` or `Data Size -` entries or icons.

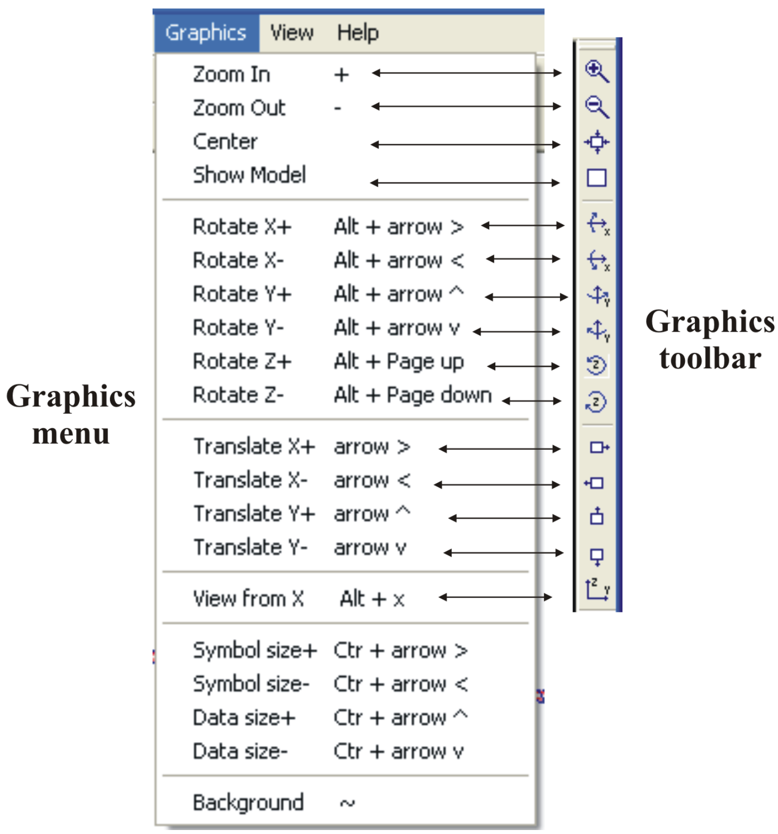

Figure 1.12: The *Graphics* menu and toolbar. Each action can be invoked by selecting a menu item or clicking the corresponding toolbar icon. In this figure, the menu items and corresponding toolbar icons are shown next to each other to highlight the correspondence..

1.6 Installation of SectionBuilder

1.6.1 Directory structure

A typical installation of the code features the following five sub-directories.

- `SectionBuilder\bin`. This directory contains the executable, `SecBuild.exe` and the input file icon `seb.ico`.

- `SectionBuilder\Demos`. This directory contains a number of sample input files illustrating the various types of cross-sections that can be defined.
- `SectionBuilder\Manual`. This directory contains the user's manual in pdf format. The same document can be found online manual
- `SectionBuilder\Materials`. This directory contains materials properties for a number of materials in both international system of units and US units. These files should not be edited.
- `SectionBuilder\Templates`. This directory contains template files that should not be edited.

1.6.2 Input files

The following file types are input files for `SectionBuilder`.

- `.sbf`. These are input files to be executed by `SectionBuilder`. The input files contain the data in the format described in this manual.
- `.tpl`. These are input files are template files that should not be edited.

1.6.3 Output files

The following file types are created by `SectionBuilder` and contain the results of the analysis.

- `.bak`. This file is backup file that duplicates the .seb input file.
- `.html`. This file echoes the definition of the model in hyper linked format.
- `.out`. This file echoes the definition of the cross-section and lists potential error and warning messages. It can be viewed using any text editor.
- `.sbp`. This file contains the sectional stiffness and mass properties computed by `SectionBuilder`. This file can be viewed using any text editor.
- `.sbs`. This file contains the three-dimensional warping displacements, stress components and strain components computed by `SectionBuilder` for various loading cases. This file can be viewed using any text editor.
- `.sva`. This file contains the information pertaining to the meshing of the cross-section. This file should not be edited.

1.6.4 Installation Procedure

Follow the installation procedure for Windows XP.

1. Extract file `SectionBuilder.rar` to `C:\`.
2. Associate the extension `.seb` with the execution of the executable `SectionBuilder\bin\SecBuild.exe`. From a folder's window, click `Tools->Folder Options`. Once the `Folder Options` dialog is open, click tab `File Types`. Click the `New` icon to `Create New Extension` dialog; type `seb` and click `OK`. With the new file type `seb` highlighted in the `Registered file types` list, click `Advanced` to open the `Edit File Type` dialog. Click `Change icon` then `Browse` and point to icon `SectionBuilder\bin\seb.ico`. Next, click on `New`; type `open` under `Action:`, then click `Browse` and point to executable `SectionBuilder\bin\SecBuild.exe` and click `OK`. Click `OK` and `Close` to exit the `Folder Options` dialog.
3. Place a short cut to `SectionBuilder\bin\SecBuild.exe` on your desktop.
4. Verify the installation. Go to `SectionBuilder\Demos\Airf` and double click on `airf1.seb`; this should open the `SectionBuilder` program.

Chapter 2

Parametric shape configurations

Parametric shape configurations have given geometric shapes that are parameterized. Parametric shape configurations are defined through the `Sections` menu item shown in fig. 4.1 and can be of the following type: airfoil sections as described in section 2.1, circular arcs as described in section 2.2, C-sections as described in section 2.3, circular cylinders as described in section 2.4, double boxes as described in section 2.5, I-sections as described in section 2.6, rectangular boxes as described in section 2.7, rectangular sections as described in section 2.8, circular tubes as described in section 2.9, triangular sections as described in section 2.10, or T-sections as described in section 2.11.

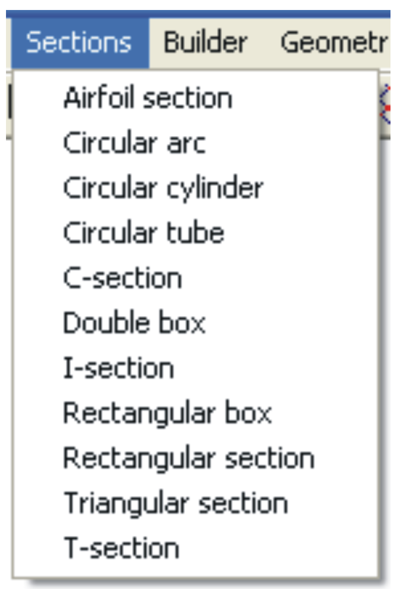

Figure 2.1: The section menu.

2.1 Definition of airfoil sections

Airfoil sections are parametric configurations with no internal web, as depicted in fig. 2.2, or with one or two internal webs, as shown in fig. 2.3 and 2.4, respectively. Airfoil sections consist of top and bottom flanges with optional internal webs. The section consists of up to three zones to which independent material properties can be assigned.

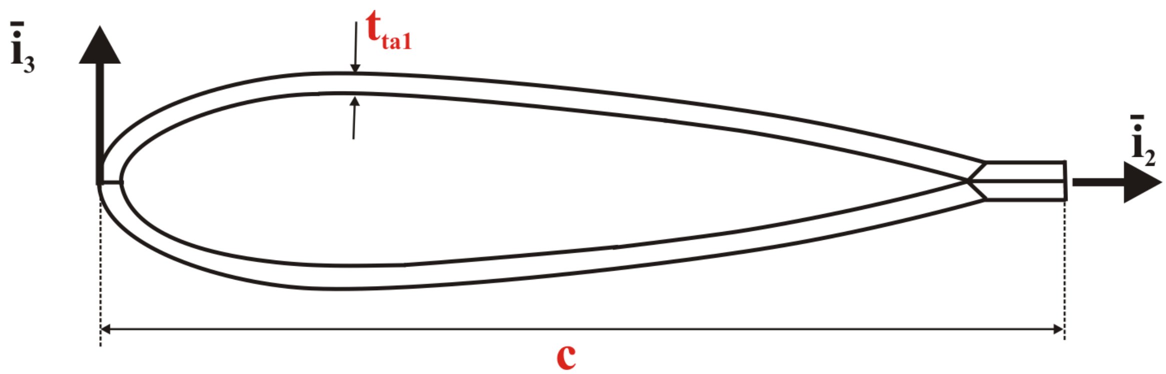

Figure 2.2: Configuration of the Airfoil Section with no web.

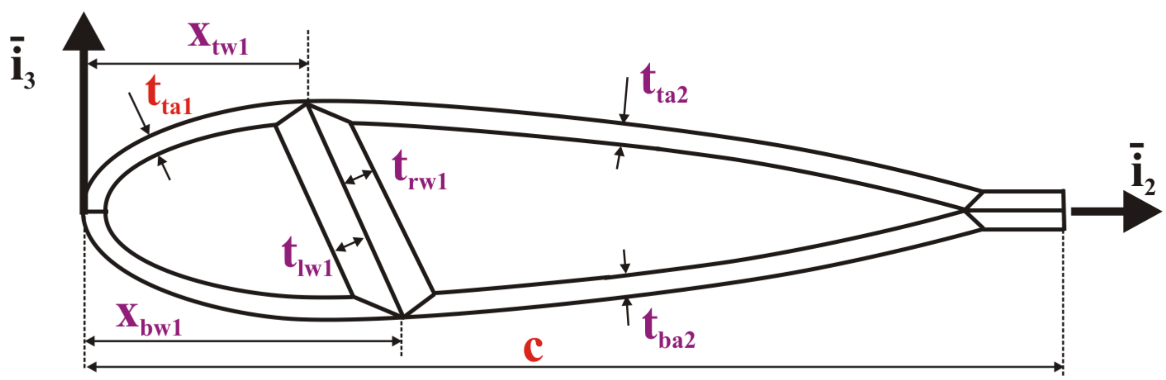

Figure 2.3: Configuration of the Airfoil Section with a single web.

Airfoil sections are defined by means of four dialog tabs.

1. The *Airfoil-section* dialog tab as described in section 2.1.1, which defines the name of the section.
2. The *Airfoil profile* dialog tab as described in section 2.1.2, which defines the outer geometry of the section as a NACA four digit series airfoil.
3. The *Dimensions* dialog tab as described in section 2.1.3, which defines the dimensions of the section.
4. The *Materials* dialog tab as described in section 2.1.4, which defines the materials the section is made of.

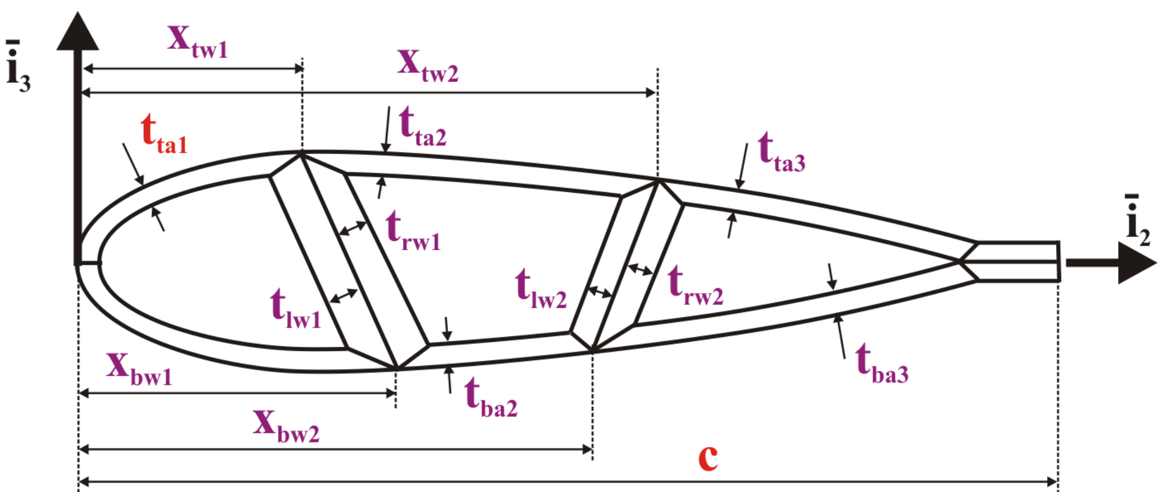

Figure 2.4: Configuration of the Airfoil Section with two webs.

2.1.1 The *Airfoil-section* dialog tab

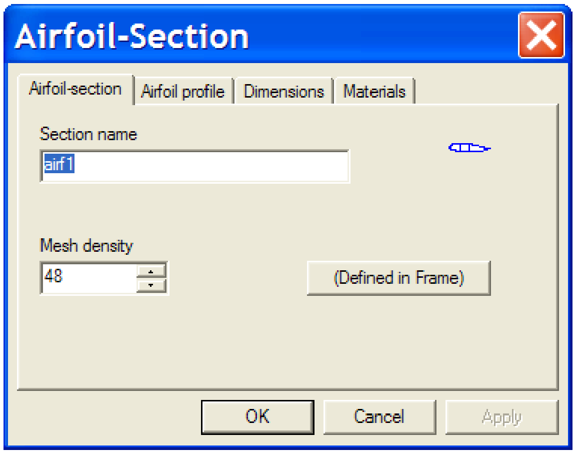

Figure 2.5: The *Airfoil-section* dialog tab.

The *Airfoil-section* dialog tab as described in fig. 2.5, defines the following data for the airfoil section.

1. **Section name.** Enter a unique name for the airfoil section.
2. **Mesh density.** Enter the desired mesh density as described in section 5.1, for the finite element discretization.
3. *(Defined in frame).* The geometry of the airfoil section can be defined with respect to a fixed frame as described in section 7.1, allowing translation and rotation of the section as a rigid body.

2.1.2 The *Airfoil profile* dialog tab

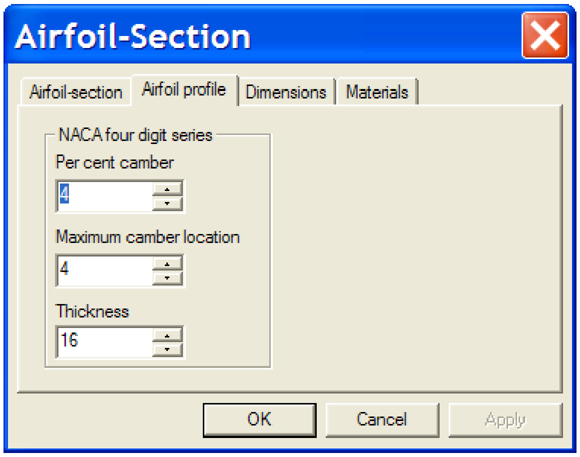

Figure 2.6: The *Airfoil profile* dialog tab.

The *Airfoil profile* dialog tab defines the outer aerodynamic profile for the airfoil section. This profile is assumed to be one of the profiles defined by the NACA four digit series [1].

1. The first digit, 1, indicates the airfoil camber in percent of the chord (**required input**: integer $1 \in [0, 9]$).

2. The second digit, 2, indicates the distance from the leading edge to the location of the maximum camber, in tenth of the chord (**required input**: $2 \in [0, 9]$).

3. The las two digits, 3, indicate the airfoil thickness in percent chord (**required input**: $3 \in [00, 09]$).

For instance, the NACA 2415 airfoil section has 2 percent camber at 0.4 of the chord from the leading edge and is 15 percent thick.

2.1.3 The *Dimensions* dialog tab

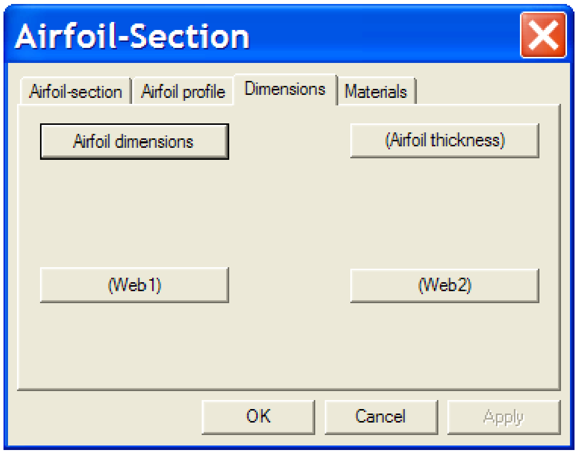

Figure 2.7: The *Dimensions* dialog tab.

The *Dimensions* dialog tab as described in fig. 2.7, defines the dimensions of the airfoil section. The section can feature no web as described in fig. 2.2, a single web as described in fig. 2.3, or two webs as described in fig. 2.4. The dimensions of the section are defined by the following parameters.

Airfoil Dimensions

1. The chord, c, of the airfoil section (**required input**).
2. The thickness, t_{ta1}, of the section wall for the no web design (**required input**). For the single or dual web design, this is the wall thickness for the front portion of the airfoil.

Airfoil Thickness

1. The top wall thickness, t_{ta2}, of the aft or middle portion of the airfoil for the single or dual web design, respectively (*default value:* $t_{\mathrm{ta2}} = t_{\mathrm{ta1}}$).
2. The bottom wall thickness, t_{ba2}, of the aft or middle portion of the airfoil for the single or dual web design, respectively (*default value:* $t_{\mathrm{ba2}} = t_{\mathrm{ta2}}$).
3. The top wall thickness, t_{ta3}, of the aft portion of the airfoil for the dual web design (*default value:* $t_{\mathrm{ta3}} = t_{\mathrm{ta2}}$).
4. The bottom wall thickness, t_{ba3}, of the aft portion of the airfoil for the dual web design (*default value:* $t_{\mathrm{ba3}} = t_{\mathrm{ta2}}$).

Web 1 Dimensions

1. The location, x_{tw1}, of the intersection of the first web with the upper airfoil profile; this is a dimensional quantity (*default value:* $x_{\mathrm{tw1}} = 0$). This variable is also used as a flag for the presence of the first web: if $x_{\mathrm{tw1}} \neq 0$, at least one web is present.
2. The location, x_{bw1}, of the intersection of the first web with the lower airfoil profile; this is a dimensional quantity (*default value:* $x_{\mathrm{bw1}} = x_{\mathrm{tw1}}$).
3. The thickness, t_{lw1}, of the left portion of the first web (*default value:* $t_{\mathrm{lw1}} = t_{\mathrm{ta1}}$).
4. The thickness, t_{rw1}, of the right portion of the first web (*default value:* $t_{\mathrm{rw1}} = t_{\mathrm{ta2}}$).

Web 2 Dimensions

1. The location, x_{tw2}, of the intersection of the second web with the upper airfoil profile; this is a dimensional quantity (*default value:* $x_{\mathrm{tw2}} = 0$). This variable is also used as a flag for the presence of the second web: if $x_{\mathrm{tw2}} \neq 0$, two webs are present.

2. The location, x_{bw2}, of the intersection of the second web with the lower airfoil profile; this is a dimensional quantity (*default value:* $x_{\mathrm{bw2}} = x_{\mathrm{tw2}}$).

3. The thickness, t_{lw2}, of the left portion of the second web (*default value:* $t_{\mathrm{lw2}} = t_{\mathrm{ta1}}$).

4. The thickness, t_{rw2}, of the right portion of the second web (*default value:* $t_{\mathrm{rw2}} = t_{\mathrm{ta2}}$).

2.1.4 The *Materials* dialog tab

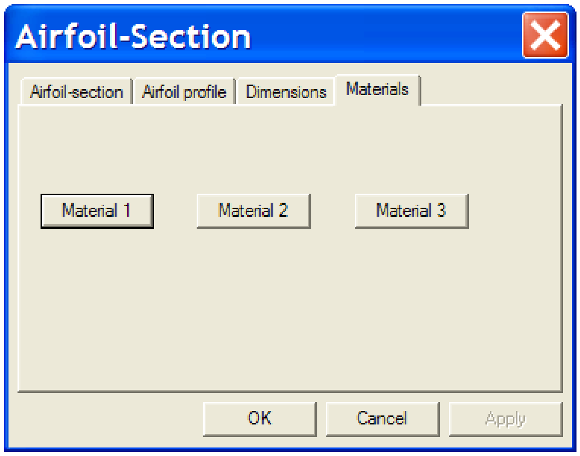

Figure 2.8: The *Materials* dialog tab.

The *Materials* dialog tab as described in fig. 2.8, defines the materials from which the airfoil section is made. As shown in fig. 2.9, the section is divided into three zones.

1. The front portion of the airfoil consists of the components labeled **Flg1U**, **Flg1L** and **LWeb1**.
2. The middle portion of the airfoil consists of the components labeled *RWeb1*, *Flg2U*, *Flg2L* and *LWeb2*; these components are present in the single and dual web designs only.
3. The aft portion of the airfoil consists of the components labeled *RWeb2*, *Flg3U*, *Flg3L*, *TedU* and *TedL*; these components are present for the dual web design only.

It is possible to assign material properties as described in section 4.1, or solid properties as described in section 4.2, to each zone of the airfoil section.

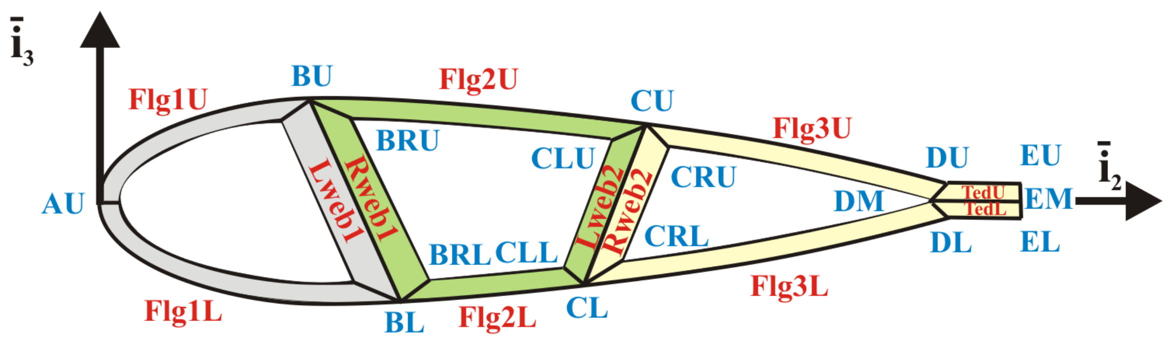

Figure 2.9: The three zones of the airfoil section.

2.1.5 Formatted input

The data that defines the airfoil section as described in the above sections will be saved in a specially formatted input file which has the following structure.

@AIRFOIL_SECTION_DEFINITION {

@AIRFOIL_SECTION_NAME { `TboxName` **} {**

@CHORD_LENGTH { c **}**
@TOP_AIRFOIL_1_THICKNESS { t_{ta1} **}**
@NACA_AIRFOIL_PROFILE { 1, n_2, n_3 **}**
@TOP_AIRFOIL_2_THICKNESS { t_{ta2} }
@BOTTOM_AIRFOIL_2_THICKNESS { t_{ba2} }
@TOP_AIRFOIL_3_THICKNESS { t_{ta3} }
@BOTTOM_AIRFOIL_3_THICKNESS { t_{ba3} }
@TOP_WEB_1_LOCATION { t_{tw1} }
@BOTTOM_WEB_1_LOCATION { t_{bw1} }
@LEFT_WEB_1_THICKNESS { t_{lw1} }
@RIGHT_WEB_1_THICKNESS { t_{rw1} }
@TOP_WEB_2_LOCATION { t_{ta3} }
@BOTTOM_WEB_2_LOCATION { t_{ba3} }
@LEFT_WEB_2_THICKNESS { t_{lw2} }
@RIGHT_WEB_2_THICKNESS { t_{rw2} }
@AIRFOIL_1_MATERIAL_NAME { `MaterialName1` }
@AIRFOIL_2_MATERIAL_NAME { `MaterialName2` }
@AIRFOIL_3_MATERIAL_NAME { `MaterialName3` }
@IS_DEFINED_IN_FRAME { `FxdFrameName` }
@MESH_DENSITY { md }

}

}

2.1.6 Examples

A few examples that describe the construction procedure of this type of section are shown below.

Example 1

This example shows a NACA 4 digit series airfoil section. The chord length, thickness, material properties and mesh density are assigned here. This example also shows the principal centroidal axes of bending.

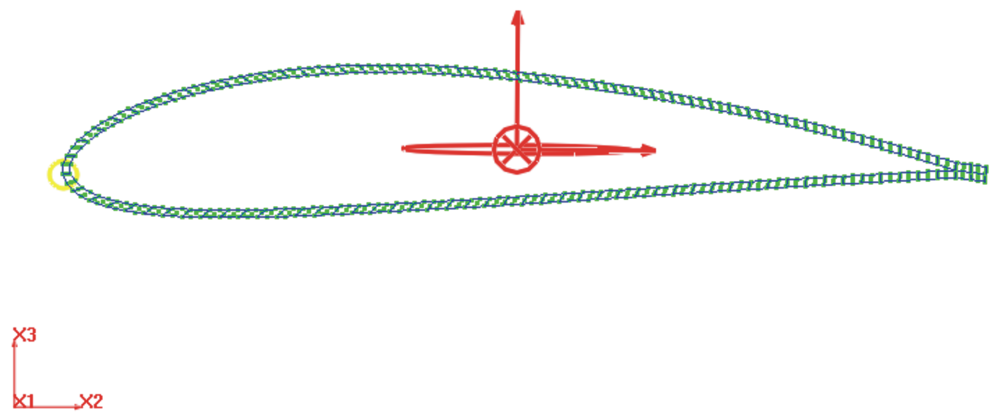

Figure 2.10: Example 1-Airfoil Section.

Example 2

This example shows a NACA 4 digit series airfoil section. The section has 2 webs. The chord length, the location of the webs, thickness of the upper, lower section and web, material properties and mesh density are assigned here. This example also shows the principal axes of inertia at the mass center.

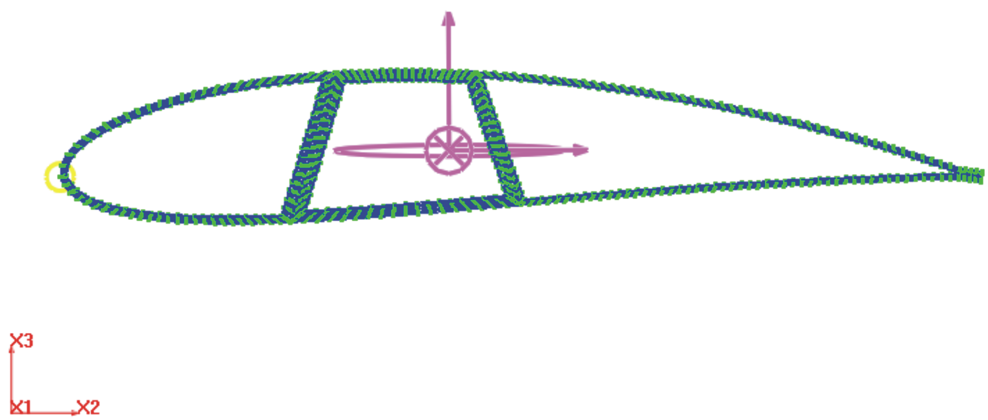

Figure 2.11: Example 2-Airfoil Section with Two Webs.

2.2 Definition of circular arcs

Circular arcs are predefined sections presenting the shape shown in fig. 2.12. Circular arcs consist of an area included between two circular arcs spanned by a common angle. The section consists of a single zone to which material properties can be assigned. The circular arc is an **open circular tube**, as shown in fig. 2.12. **Closed circular tubes** can be defined with the help of the circular tube predefined section as described in section 2.9.

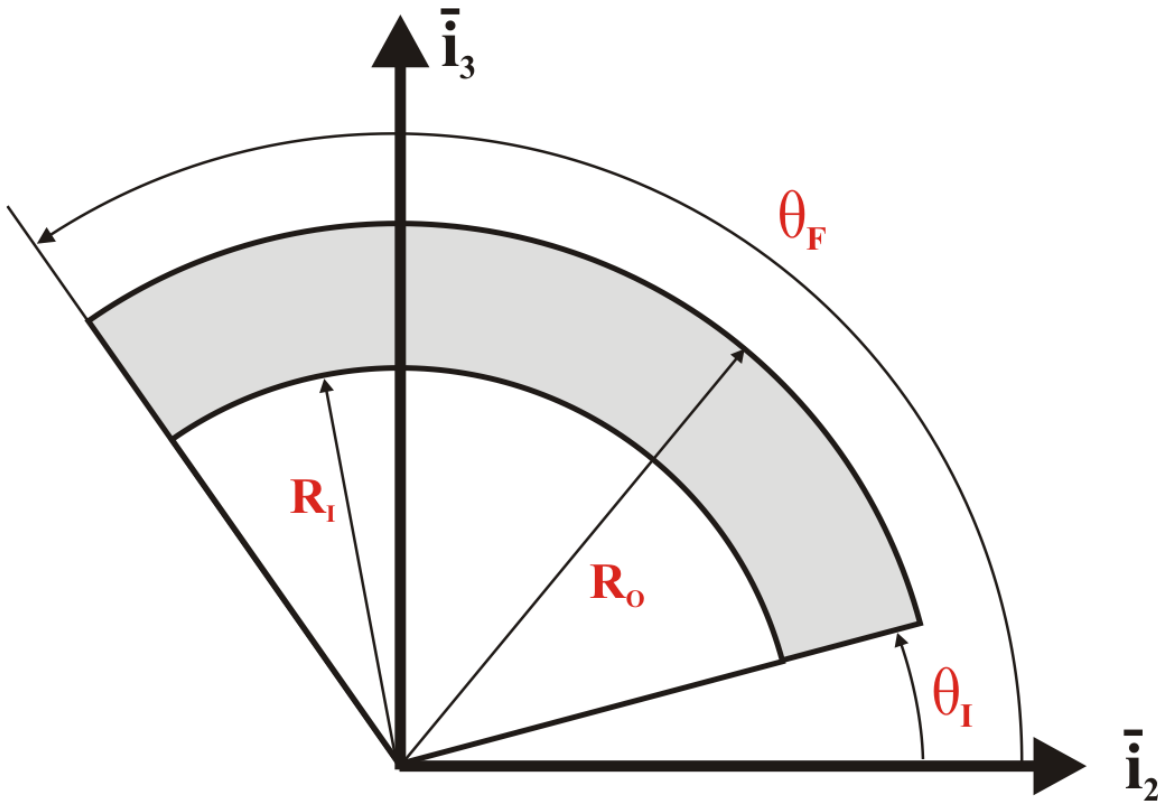

Figure 2.12: Configuration of the circular arc.

Circular arcs are defined by means of three dialog tabs.

1. The *Circular Arc* dialog tab as described in section 2.2.1, which defines the name of the section.
2. The *Dimensions* dialog tab as described in section 2.2.2, which defines the dimensions of the section.
3. The *Materials* dialog tab as described in section 2.2.3, which defines the materials the section is made of.

2.2.1 The *Circular Arc* dialog tab

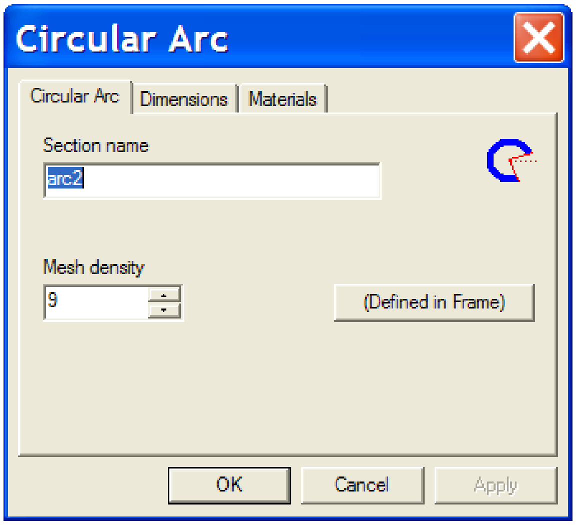

Figure 2.13: The *Circular Arc* dialog tab.

The *Circular Arc* dialog tab as described in fig. 2.13, defines the following data for the circular arc.

1. **Section name.** Enter a unique name for the circular arc.
2. **Mesh density.** Enter the desired mesh density as described in section 5.1, for the finite element discretization.
3. *(Defined in frame).* The geometry of the circular arc can be defined with respect to a fixed frame as described in section 7.1, allowing translation and rotation of the section as a rigid body.

2.2.2 The *Dimensions* dialog tab

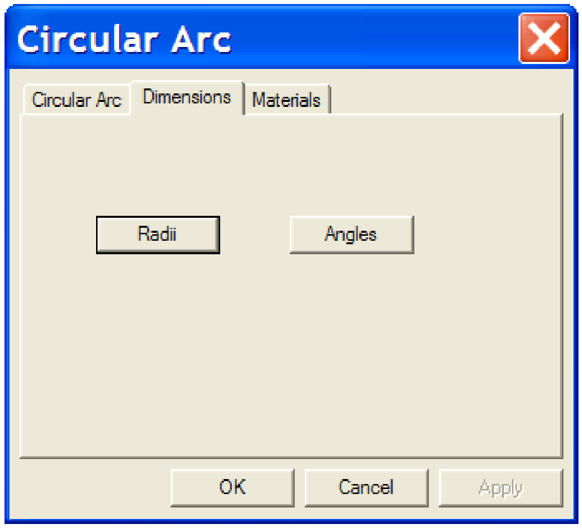

Figure 2.14: The *Dimensions* dialog tab.

The *Dimensions* dialog tab as described in fig. 2.14, defines the dimensions of the circular arc shown in fig. 2.12. The dimensions of the section are defined by the following parameters.

1. The outer radius, R_O, and inner radius, R_I, of the circular tube (**required input**).

2. The initial angle, θ_I, and final angle, θ_F, of the circular arc, both measured in degrees (**required input**). Note: $0 \leq \theta_I < 360$, $0 < \theta_F \leq 360$ and $\theta_I < \theta_F$. Even when $\theta_I = 0$ and $\theta_F = 360$, the circular arc is still an open circular tube.

2.2.3 The *Materials* dialog tab

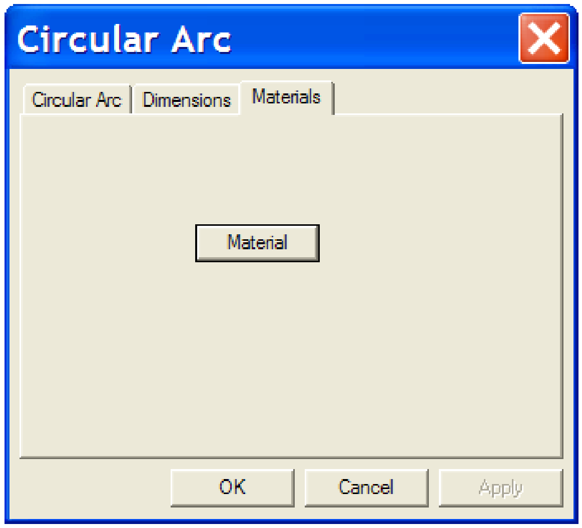

Figure 2.15: The *Materials* dialog tab.

The *Materials* dialog tab as described in fig. 2.15, defines the materials the circular arc is made of. As shown in fig. 2.12, the section consists of a single zone to which it is possible to assign material properties as described in section 4.1, or solid properties as described in section 4.2.

2.2.4 Formatted input

The data that defines the circular arc as described in the above sections will be saved in a specially formatted input file which has the following structure.

@CIRCULAR_ARC_DEFINITION {

@CIRCULAR_ARC_NAME { `TboxName` **} {**

@INNER_RADIUS { R_I **}**
@OUTER_RADIUS { R_O **}**
@THETA_INITIAL { θ_I **}**
@THETA_FINAL { θ_F **}**
@MATERIAL_PROPERTY_NAME { `MaterialName` }
@IS_DEFINED_IN_FRAME { `FxdFrameName` }
@MESH_DENSITY { md }

}

}

2.2.5 Examples

A few examples that describe the construction procedure of this type of section are shown below.

Example 1

This example shows a circular arc. Here the inner and outer radii, initial (0 degree) and final (180 degrees) angular positions of the arc, material and mesh density are assigned for constructing this section. This example also shows the principal axes of shearing at the shear center.

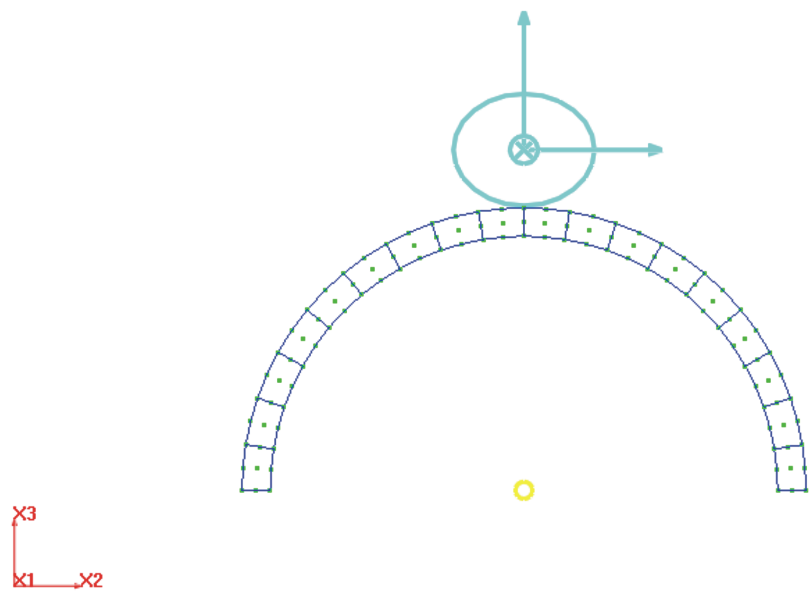

Figure 2.16: Example 2-Circular Arc Section.

Example 2

This example shows a circular arc. Here the inner and outer radii, initial and final angular positions of the arc, material and mesh density are assigned for constructing this section. This example also shows the shear strain field over the cross-section under the applied sectional loads.

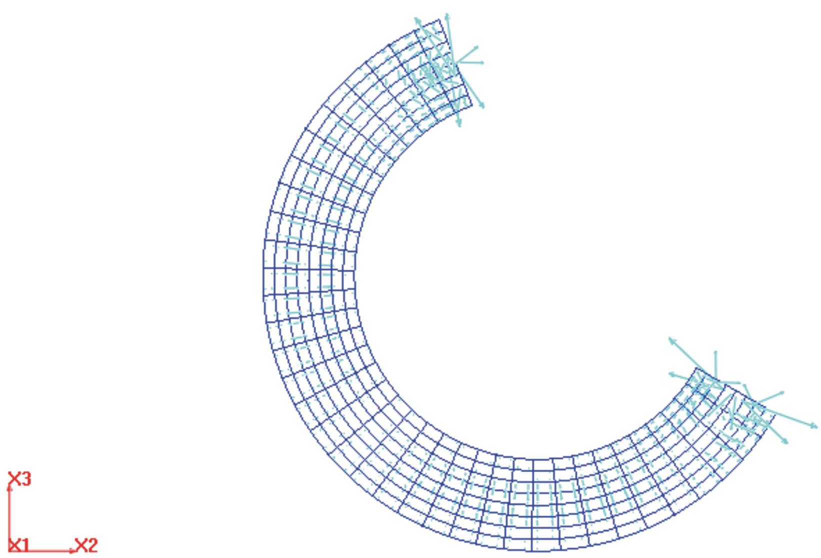

Figure 2.17: Example 2-Circular Arc Section.

2.3 Definition of C-sections

C-sections are parametric configurations of the shape depicted in fig. 2.18. They consist of a C-section, possibly reinforced by top and/or bottom flanges. The section consists of up to three zones to which independent material properties can be assigned. Note that through the use of fixed frames as described in section 7.1, the C-section can be made to look like the following shapes: ⊓, ⊔, ⊏, or ⊐.

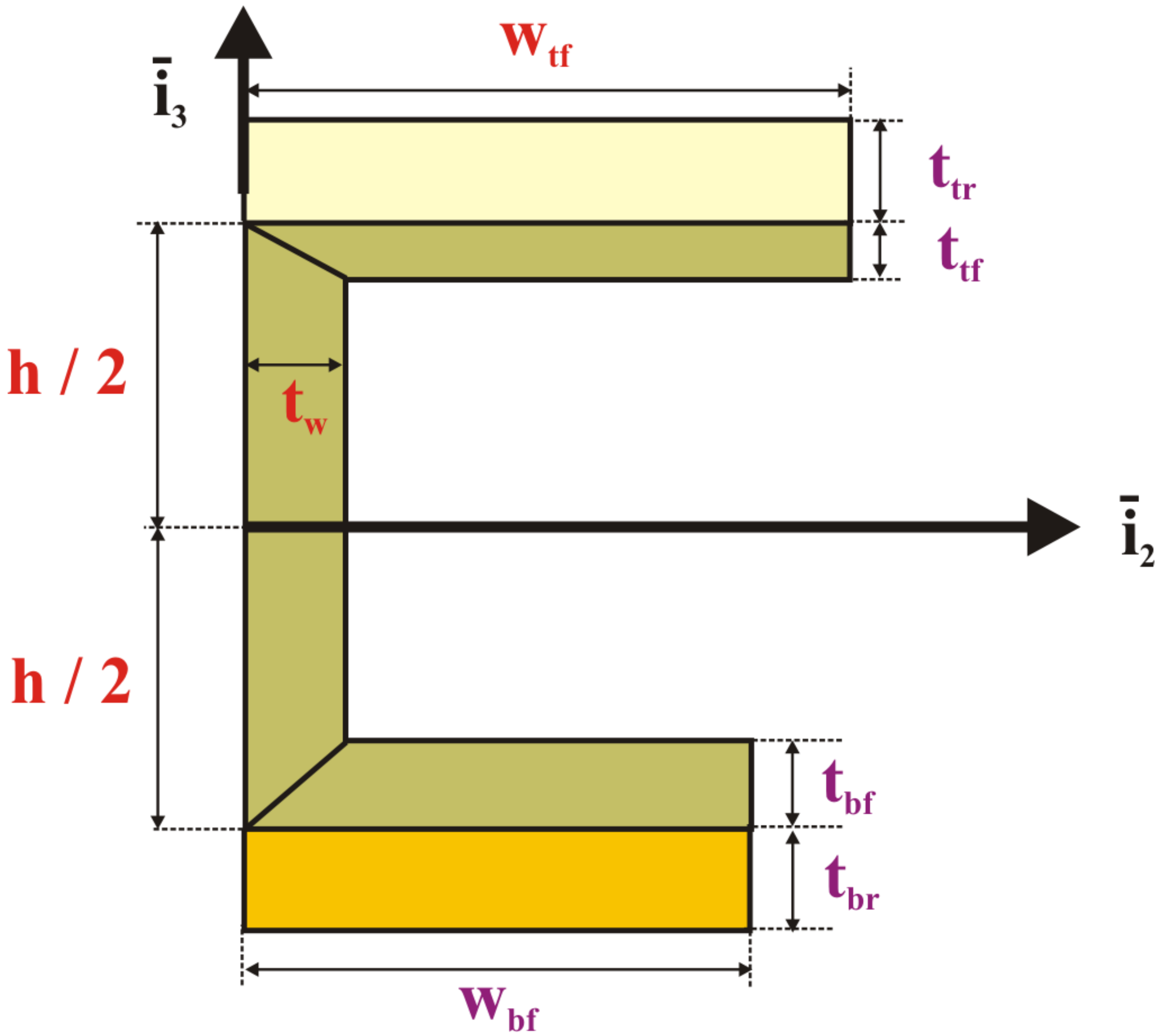

Figure 2.18: Configuration of the C-section.

C-sections are defined by means of three dialog tabs.

1. The *C-section* dialog tab as described in section 2.3.1, which defines the name of the section.
2. The *Dimensions* dialog tab as described in section 2.3.2, which defines the dimensions of the section.
3. The *Materials* dialog tab as described in section 2.3.3, which defines the materials the section is made of.

2.3.1 The *C-section* dialog tab

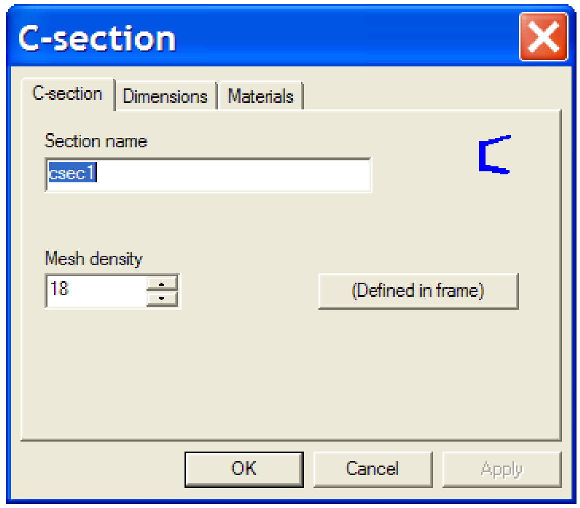

Figure 2.19: The *C-section* dialog tab.

The *C-section* dialog tab as described in fig. 2.19, defines the following data for the C-section.

1. **Section name.** Enter a unique name for the C-section.
2. **Mesh density.** Enter the desired mesh density as described in section 5.1, for the finite element discretization.
3. *(Defined in frame).* The geometry of the C-section can be defined with respect to a fixed frame as described in section 7.1, allowing translation and rotation of the section as a rigid body.

2.3.2 The *Dimensions* dialog tab

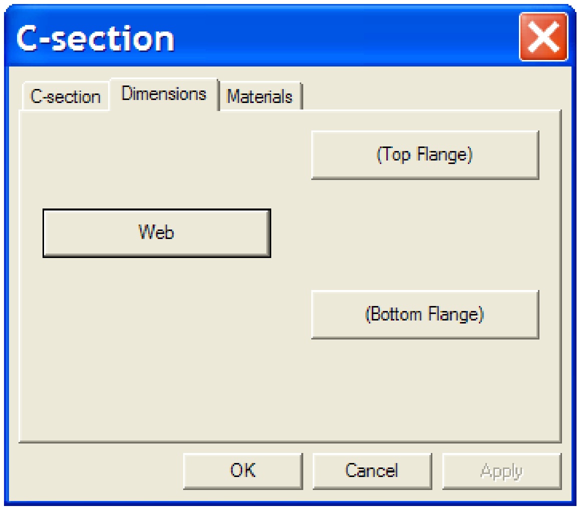

Figure 2.20: The *Dimensions* dialog tab.

The *Dimensions* dialog tab as described in fig. 2.20, defines the dimensions of the C-section, as depicted in fig. 2.18. The dimensions of the section are defined by the following parameters.

Web dimensions

1. The height, h, of the web (**required input**).
2. The thickness, t_{w}, of the web (**required input**).

Top flange dimensions

1. The width, w_{tf}, of the top flange(**required input**).
2. The thickness, t_{tf}, of the top flange (*default value:* $t_{\mathrm{tf}} = t_{\mathrm{w}}$).
3. The skew angle, α_{tf}, of the top flange, positive up, measured in degrees (*default value:* $\alpha_{\mathrm{tf}} = 0$).
4. The thickness, t_{tr}, of the top reinforcement flange; this thickness applies to both left and right reinforcements, which cannot exist independently of each other (*default value:* $t_{\mathrm{tr}} = 0$). This variable is also used as a flag for the presence of the top flange reinforcement: if $t_{\mathrm{tr}} > 0$, this reinforcement is present.

Bottom flange dimensions

1. The width, w_{bf}, of the bottom flange(*default value:* $w_{\mathrm{bf}} = w_{\mathrm{tf}}$).
2. The thickness, t_{bf}, of the bottom flange (*default value:* $t_{\mathrm{bf}} = t_{\mathrm{tf}}$).
3. The skew angle, α_{bf}, of the top flange, positive down, measured in degrees (*default value:* $\alpha_{\mathrm{bf}} = 0$).
4. The thickness, t_{br}, of the bottom reinforcement flange; this thickness applies to both left and right reinforcements, which cannot exist independently of each other (*default value:* $t_{\mathrm{br}} = 0$). This variable is also used as a flag for the presence of the bottom flange reinforcement: if $t_{\mathrm{br}} > 0$, this reinforcement is present.

2.3.3 The *Materials* dialog tab

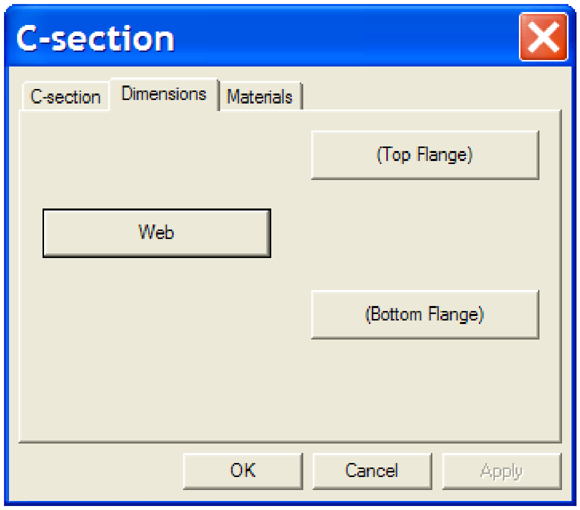

Figure 2.21: The *Materials* dialog tab.

The *Materials* dialog tab as described in fig. 2.21, defines the materials the C-section is made of. As shown in fig. 2.22, the section is divided into three zones.

1. The top reinforcement flange consists of a single solid labeled *TFlgR*.
2. The C-section it self consists of the solids labeled **TFlg**, **Web**, **BFlg**.
3. The bottom reinforcement flange consists of a single solid labeled *BFlgR*.

It is possible to assign material properties as described in section 4.1, or solid properties as described in section 4.2, to each zone of the C-section.

Special Cases

1. If the width, $w_{\mathrm{tf}} = 0$, the section looks like L-shape depicted in fig. 2.23. This L-shape section does not have TFlgR and TFlg solids shown in fig. 2.24.
2. If the width, $w_{\mathrm{bf}} = 0$, the section looks like reverse L-shape depicted in fig. 2.25. This reverse L-shape section does not have BFlgR and BFlg solids shown in fig. 2.26.
3. If the widths, $w_{\mathrm{tf}} = 0$ and $w_{\mathrm{bf}} = 0$, the section looks like a strip depicted in fig. 2.27. This strip section does not have TFlgR, TFlg, BFlgR and BFlg solids shown in fig. 2.28. It only has Web.

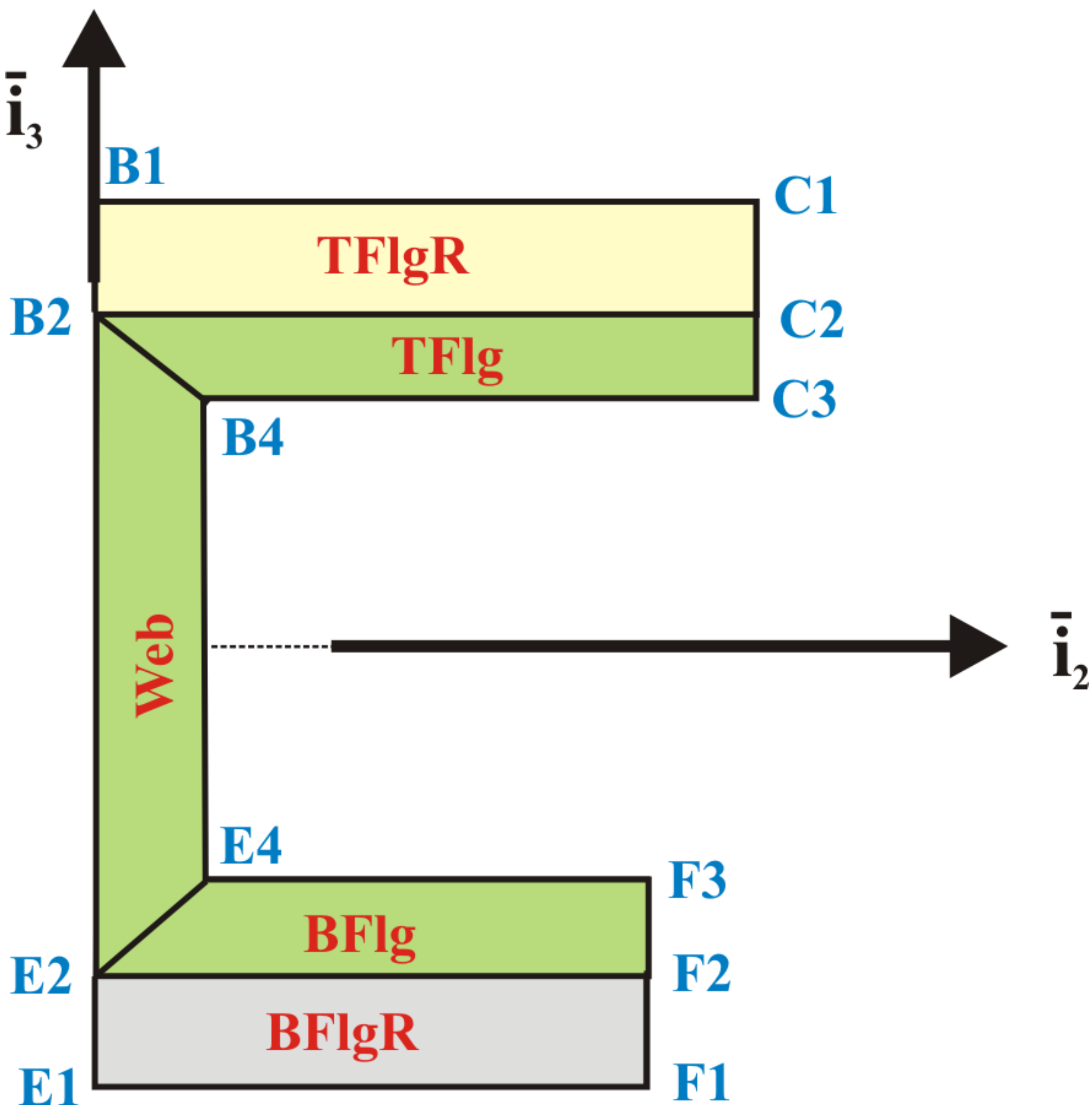

Figure 2.22: The three zones of the C-section.

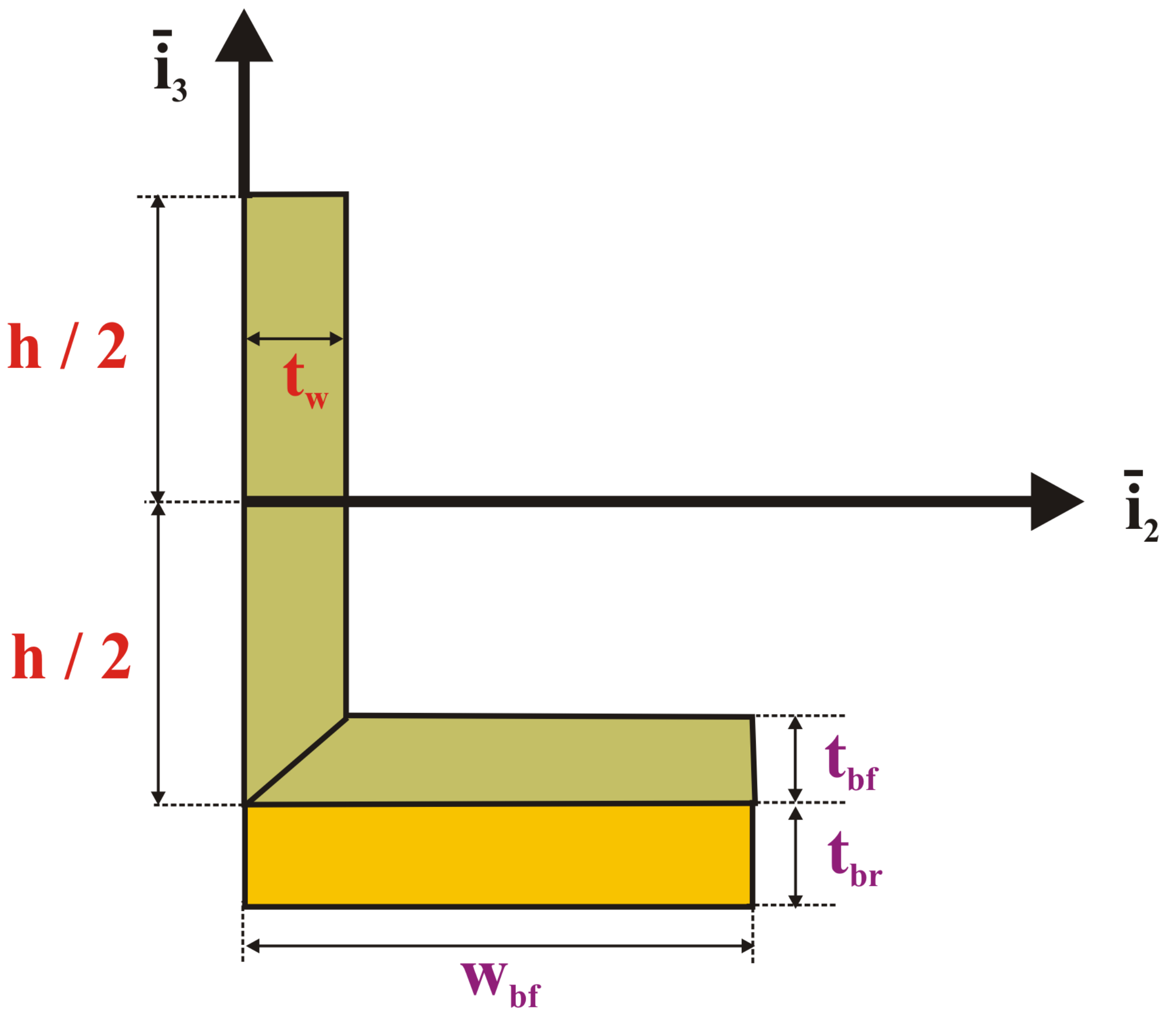

Figure 2.23: Configuration of the L-section.

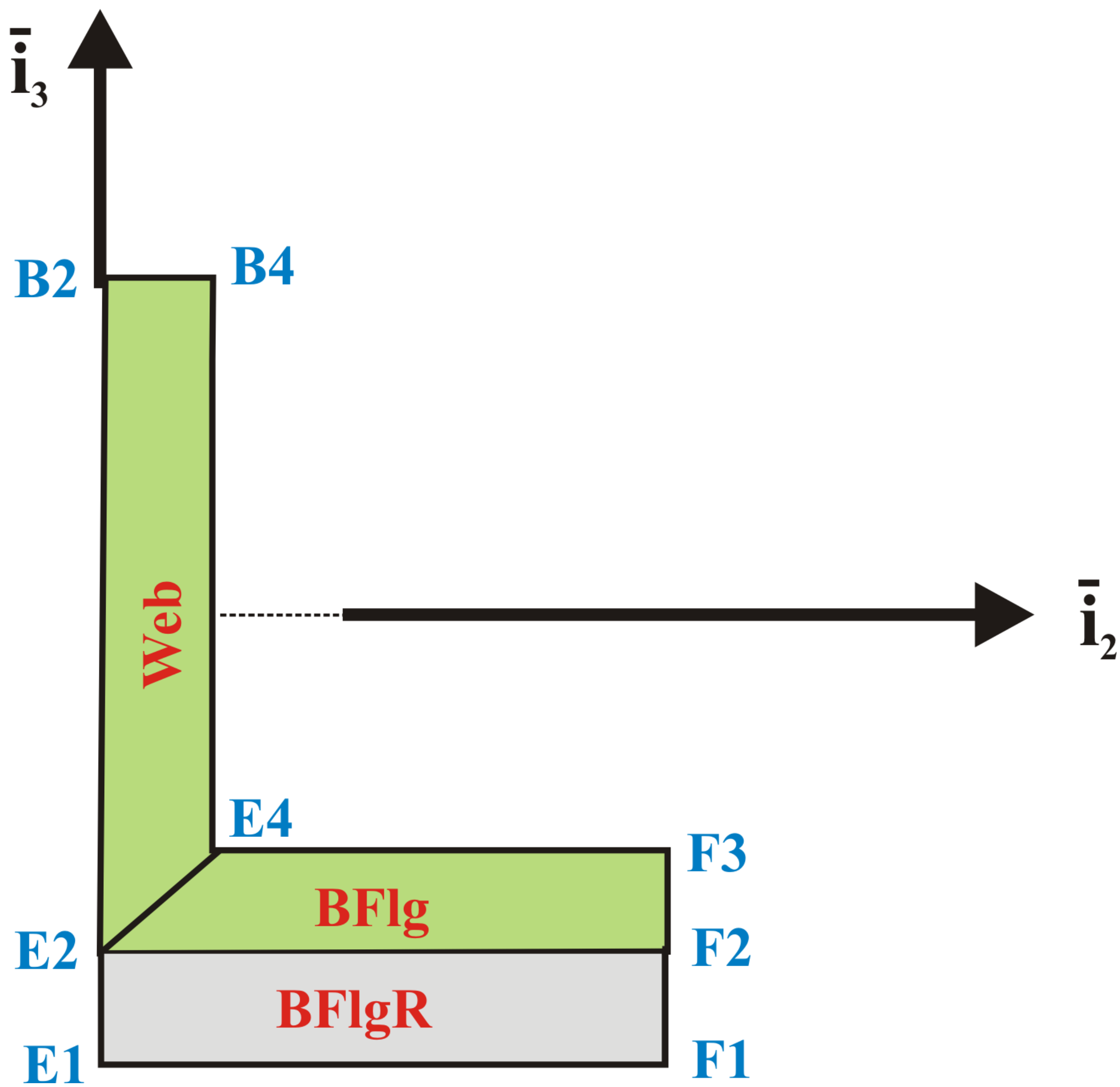

Figure 2.24: The two zones of the L-section.

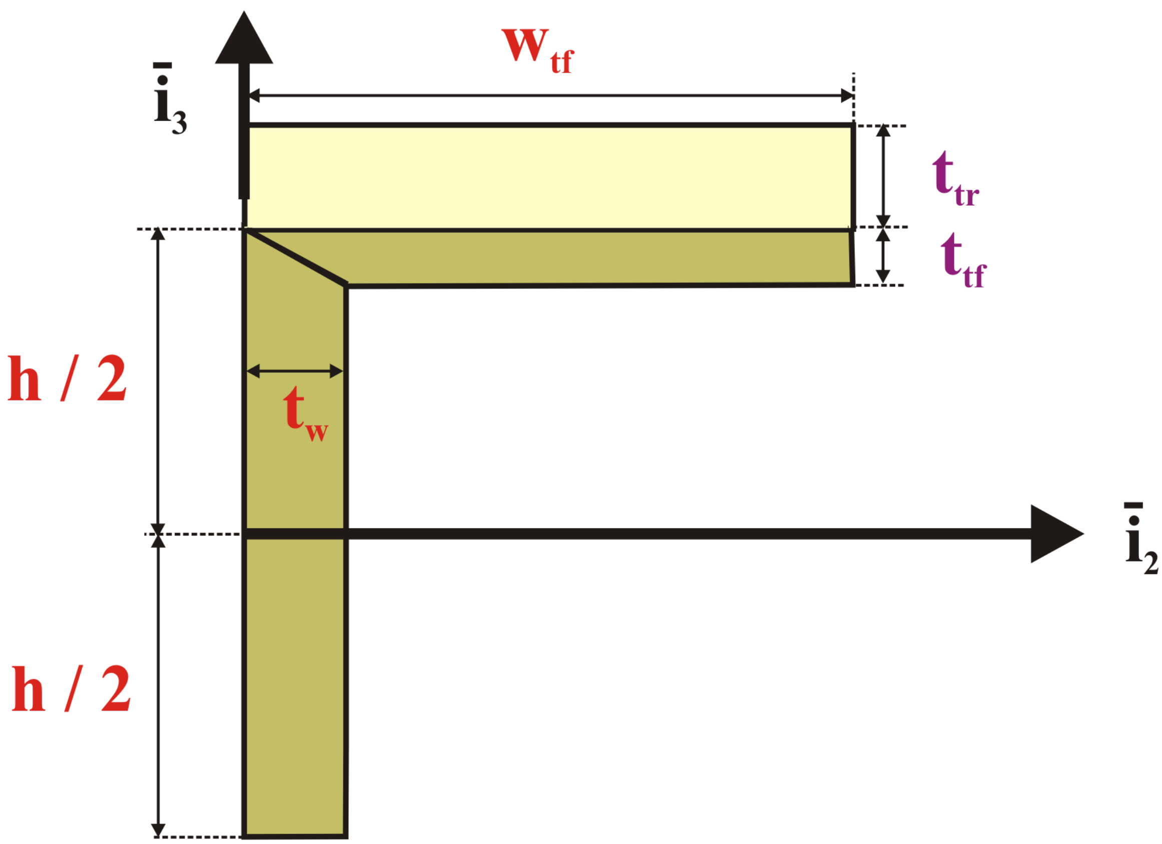

Figure 2.25: Configuration of the reverse L-section.

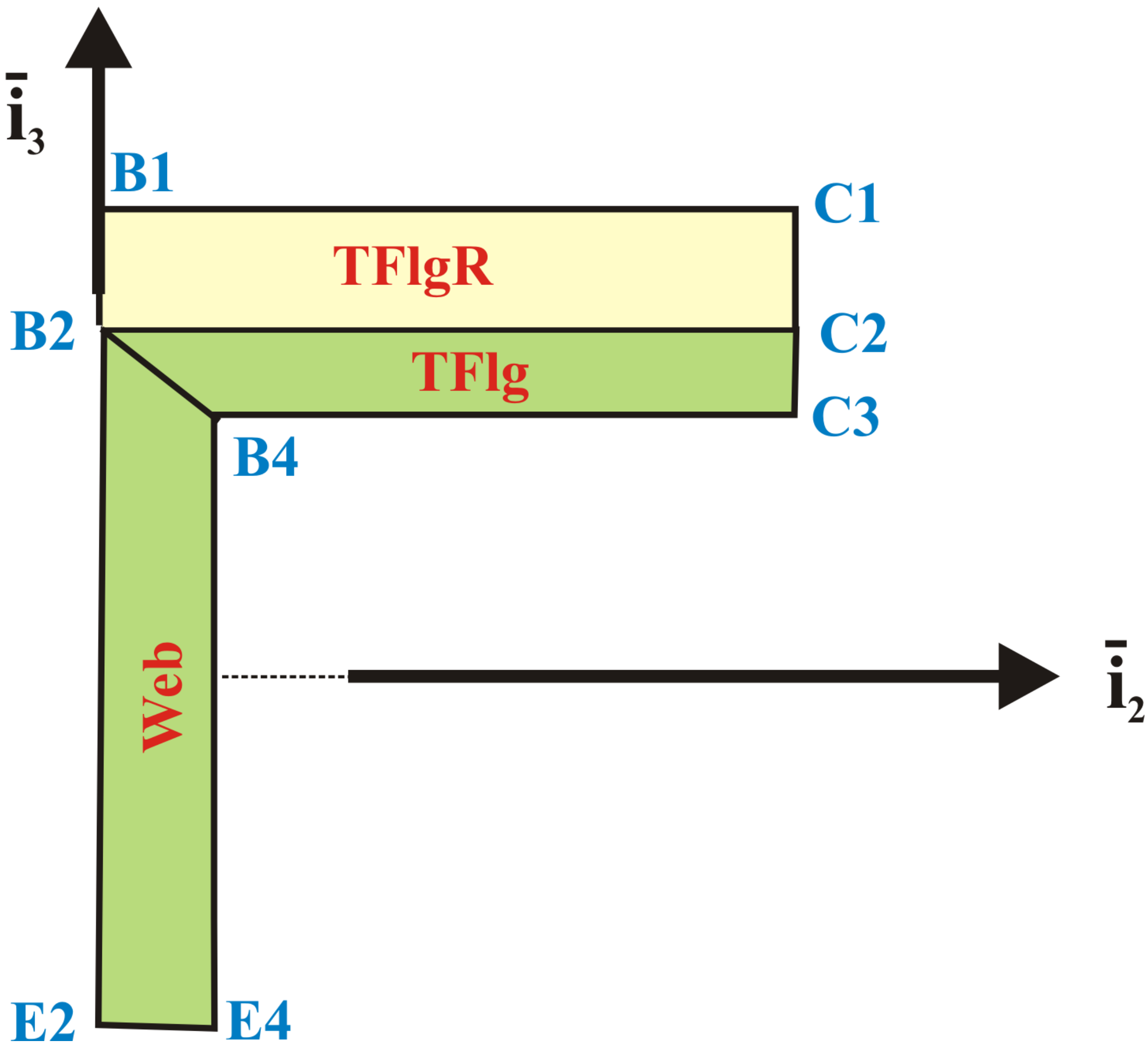

Figure 2.26: The two zones of the reverse L-section.

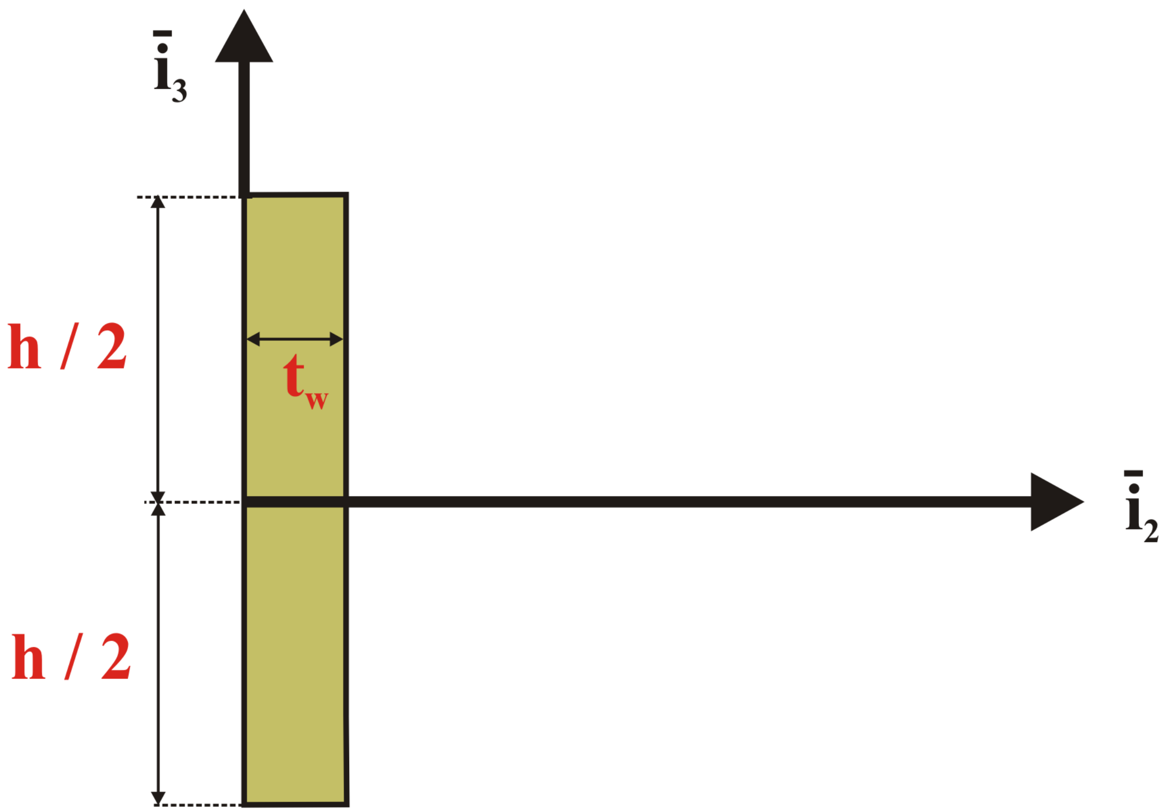

Figure 2.27: Configuration of the strip section.

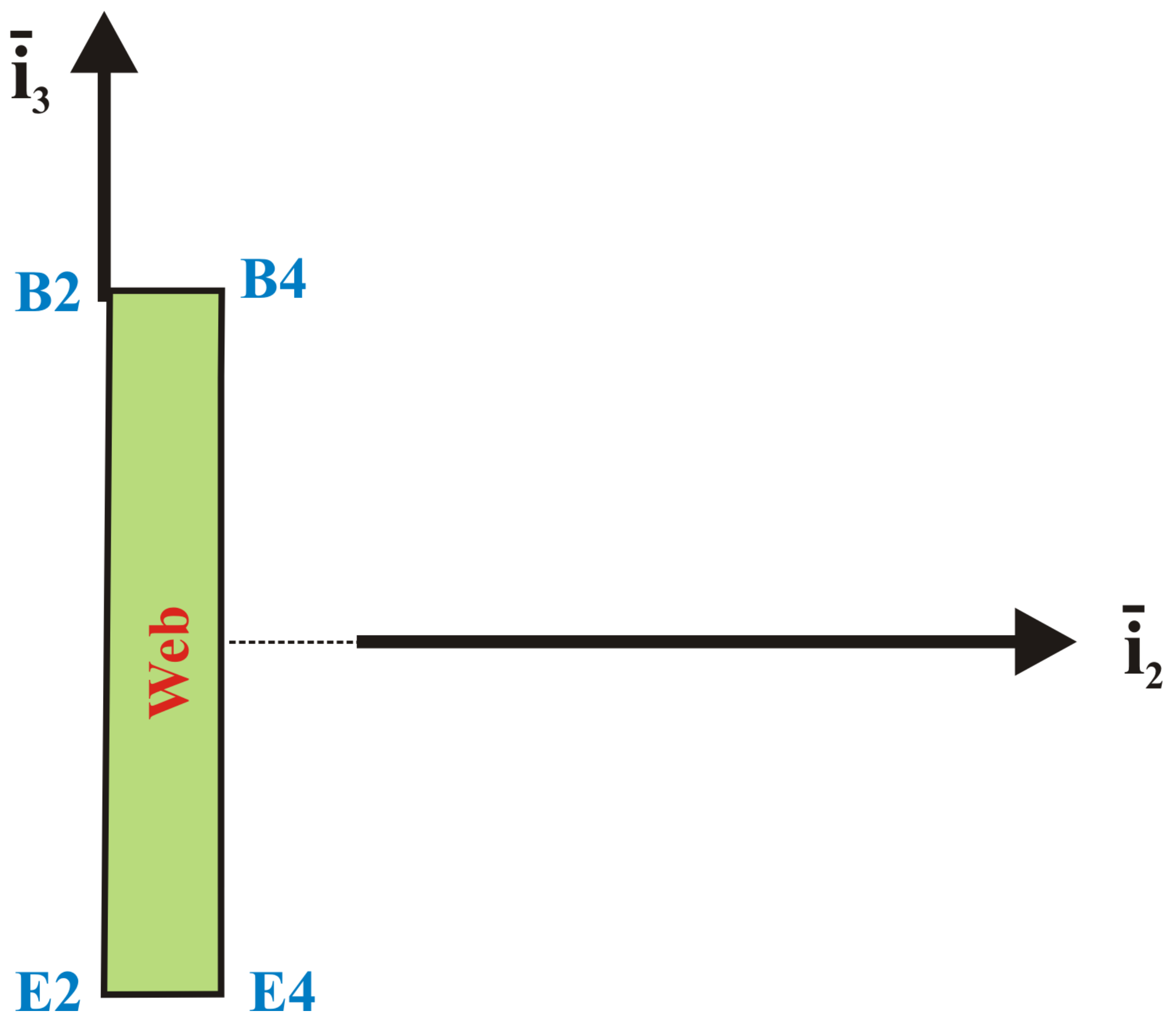

Figure 2.28: The zone of the strip section.

2.3.4 Formatted input

The data that defines the C-section as described in the above sections will be saved in a specially formatted input file which has the following structure.

@C_SECTION_DEFINITION {

@C_SECTION_NAME { `CsecName` **} {**

@WEB_HEIGHT { h **}**
@WEB_THICKNESS { t_{w} **}**
@TOP_FLANGE_WIDTH { w_{tf} **}**
@TOP_FLANGE_THICKNESS { t_{tf} }
@TOP_FLANGE_SKEW_ANGLE { α_{tf} }
@TOP_FLANGE_REINFORCE_THICKNESS { t_{tr} }
@BOTTOM_FLANGE_WIDTH { w_{bf} }
@BOTTOM_FLANGE_THICKNESS { t_{bf} }
@BOTTOM_FLANGE_SKEW_ANGLE { α_{bf} }
@BOTTOM_FLANGE_REINFORCE_THICKNESS { t_{br} }
@WEB_MATERIAL_NAME { `WebMaterialName` }
@TOP_REINFORCE_MATERIAL_NAME { `TopMaterialName` }
@BOTTOM_REINFORCE_MATERIAL_NAME { `BottomMaterialName` }
@IS_DEFINED_IN_FRAME { `FxdFrameName` }
@MESH_DENSITY { md }

}

}

2.3.5 Examples

A few examples that describe the construction procedure of this type of section are shown below.

Example 1

This example shows a C-section. Here web height, web thickness, top flange width, web material and mesh density are assigned for constructing this section. This example also shows the shear strain field over the cross-section under the applied sectional loads.

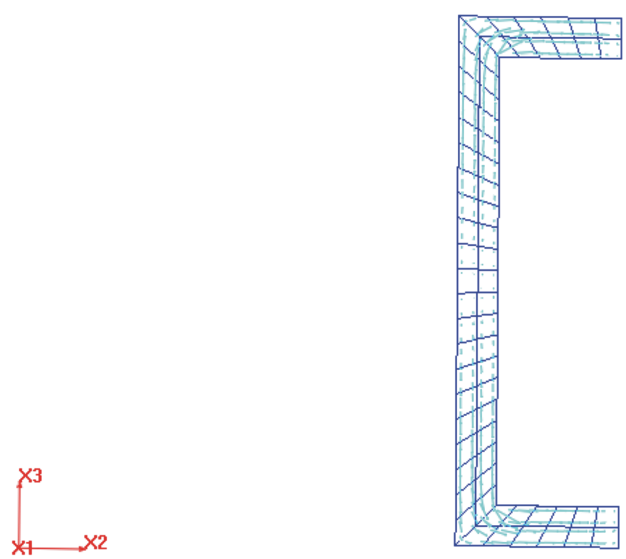

Figure 2.29: Example 1.C-Section

Example 2

This example shows also a C-section with skewed top and bottom flanges. Here web height, web thickness, top flange width, top flange thickness, flange reinforce thicknesses, flange skew angles, materials and mesh density are assigned for constructing this section. This example also shows the axial stress field over the cross-section under the applied sectional loads.

Example 3

This example shows a reverse L-section. Here web height, web thickness, top flange width, bottom flange width=0, web material and mesh density are assigned for constructing this section. This example also shows principal centroidal axes of bending.

Example 4

This example shows a L-section. Here web height, web thickness, top flange width=0, bottom flange width, flange reinforce thickness, materials and mesh density are assigned for constructing this section. This example also shows axial stress field over the cross-section under the applied sectional loads.

Example 5

This example shows a strip section. Here web height, web thickness, top flange width = 0, bottom flange width=0, web material and mesh density are assigned for constructing this section. This example also shows principal axes of inertia at the mass center.

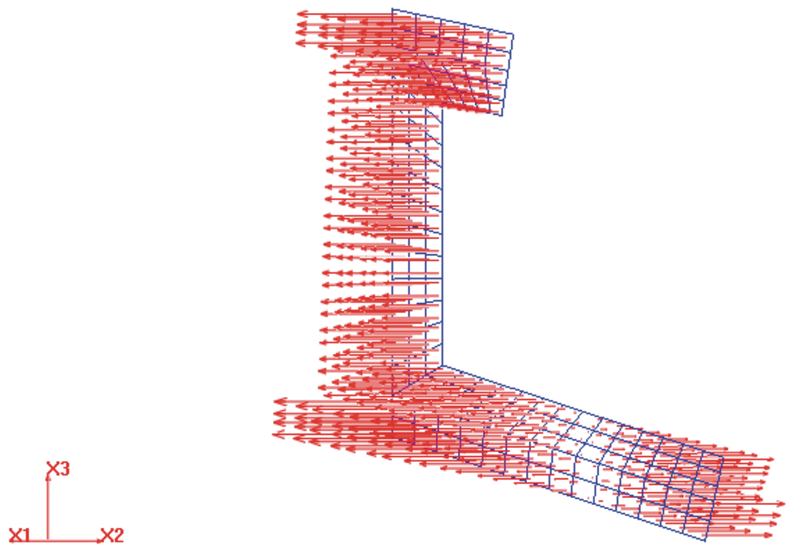

Figure 2.30: Example 2.C-Section

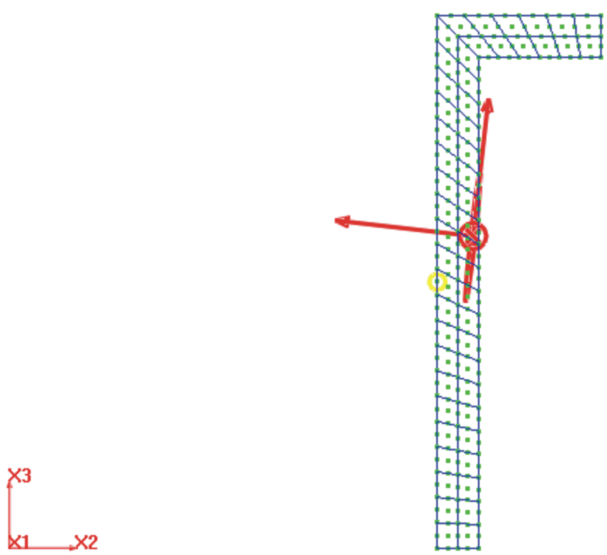

Figure 2.31: Example 3.Reverse L-Section

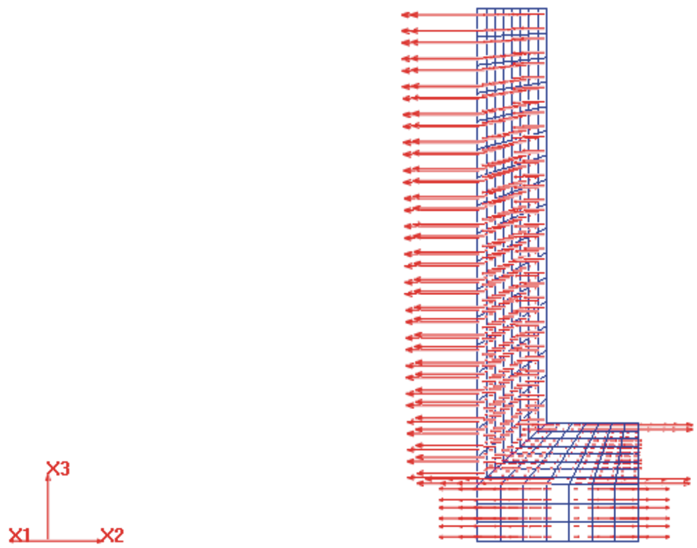

Figure 2.32: Example 4. L-Section

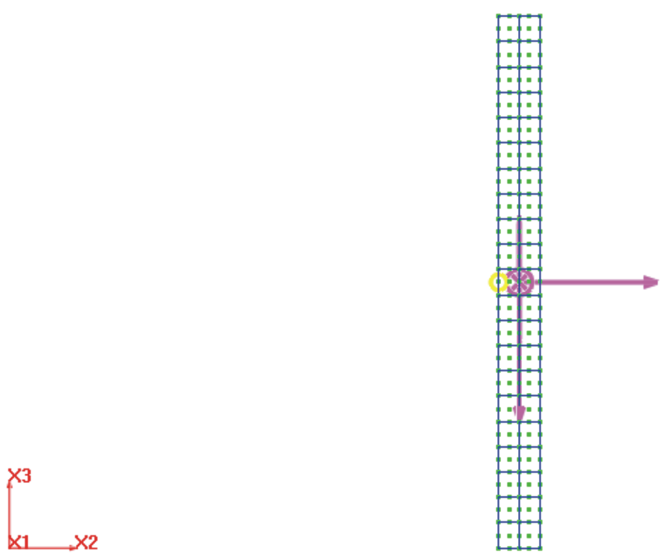

Figure 2.33: Example 5.Strip Section

2.4 Definition of circular cylinders

Circular cylinders are predefined sections presenting the shape shown in fig. 2.34. Circular cylinders consist of a solid circular cylinder. The section consists of a single zone to which material properties can be assigned. The circular cylinders are solid cylinders, as shown in fig. 2.34. Hollow circular cylinder, or circular tubes can be defined with the help of the circular tube predefined section as described in section 2.9.

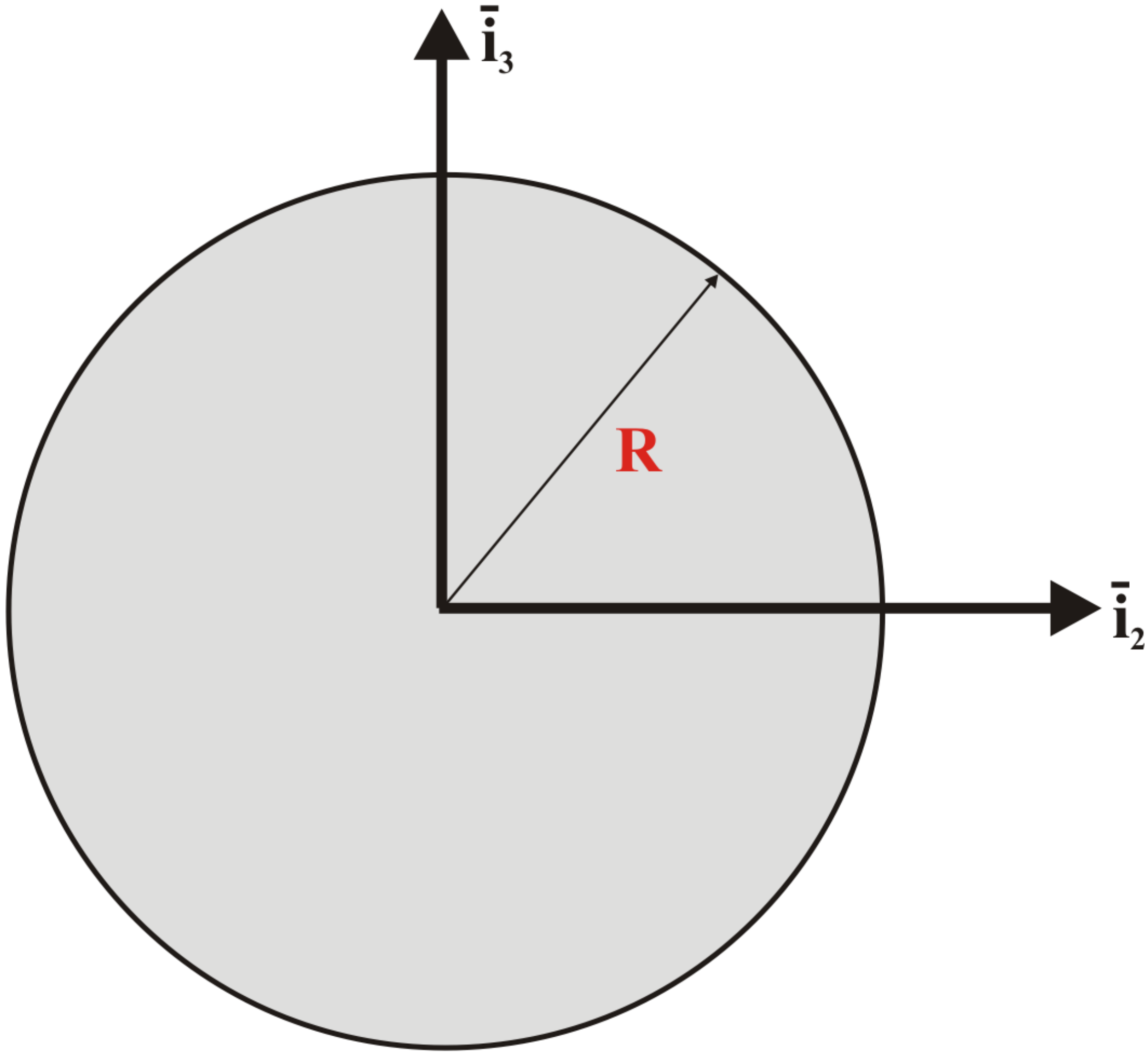

Figure 2.34: Configuration of the circular cylinder.

Circular cylinders are defined by means of three dialog tabs.

1. The *Cylinder* dialog tab as described in section 2.4.1, which defines the name of the section.
2. The *Dimensions* dialog tab as described in section 2.4.2, which defines the dimensions of the section.
3. The *Materials* dialog tab as described in section 2.4.3, which defines the materials the section is made of.

2.4.1 The *Cylinder* dialog tab

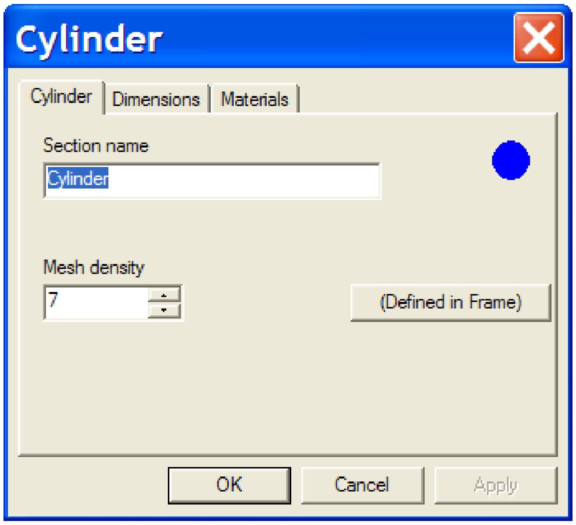

Figure 2.35: The *Cylinder* dialog tab.

The *Cylinder* dialog tab as described in fig. 2.35, defines the following data for the circular cylinder.

1. **Section name.** Enter a unique name for the circular cylinder.

2. **Mesh density.** Enter the desired mesh density as described in section 5.1, for the finite element discretization.

3. *(Defined in frame).* The geometry of the circular cylinder can be defined with respect to a fixed frame as described in section 7.1, allowing translation and rotation of the section as a rigid body.

2.4.2 The *Dimensions* dialog tab

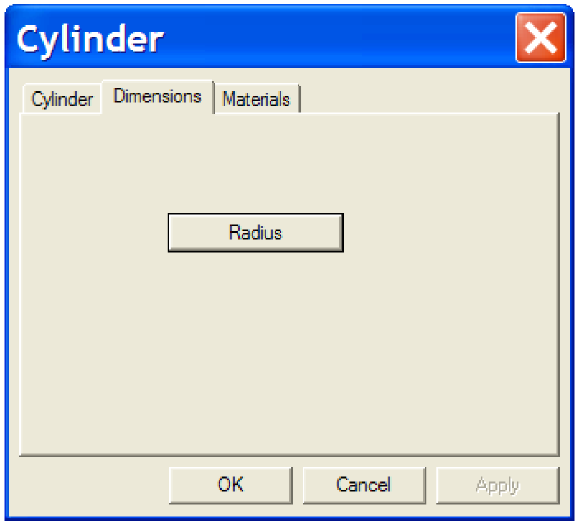

Figure 2.36: The *Dimensions* dialog tab.

The *Dimensions* dialog tab defines the dimensions of the circular cylinder shown in fig. 2.34. The dimensions of the section are defined by a single parameter.

1. The radius, R of the cylinder (**required input**).

2.4.3 The *Materials* dialog tab

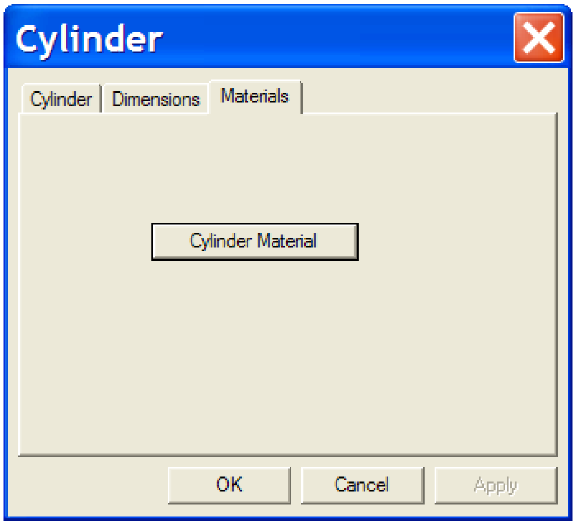

Figure 2.37: The *Materials* dialog tab.

The *Materials* dialog tab defines the materials the circular cylinder is made of. The section is made of a single, homogeneous material. It is possible to assign material properties as described in section 4.1, to the section.

2.4.4 Formatted input

The data defined in the above sections will be saved in a specially formatted input file which has the following structure.

@CYLINDER_DEFINITION {

@CYLINDER_NAME { `CyldName` **} {**

@RADIUS { R **}**

@MATERIAL_PROPERTY_NAME { `MaterialName` }

@IS_DEFINED_IN_FRAME { `FxdFrameName` }

@MESH_DENSITY { md }

}

}

2.4.5 Examples

An example that describes the construction procedure of this type of section is shown below.

Example 1

This example shows a Circular cylinder. Here radius, frame definition, material and mesh density are assigned for constructing this section. This example also shows the warping displacement of the cross-section under the applied sectional loads.

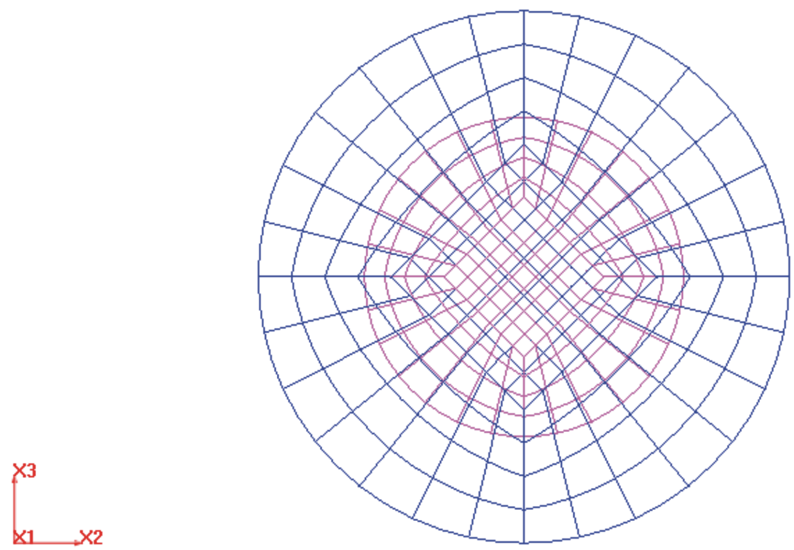

Figure 2.38: Example 1-Cylinder.

2.5 Definition of double boxes

Double boxes are parametric configurations of the shape depicted in fig. 2.39. They consist of an double box, possibly reinforced by top and/or bottom flanges. The section consists of up to four zones to which independent material properties can be assigned. Note that through the use of fixed frames as described in section 7.1, the double box can be made to look like the following shapes: H-sections.

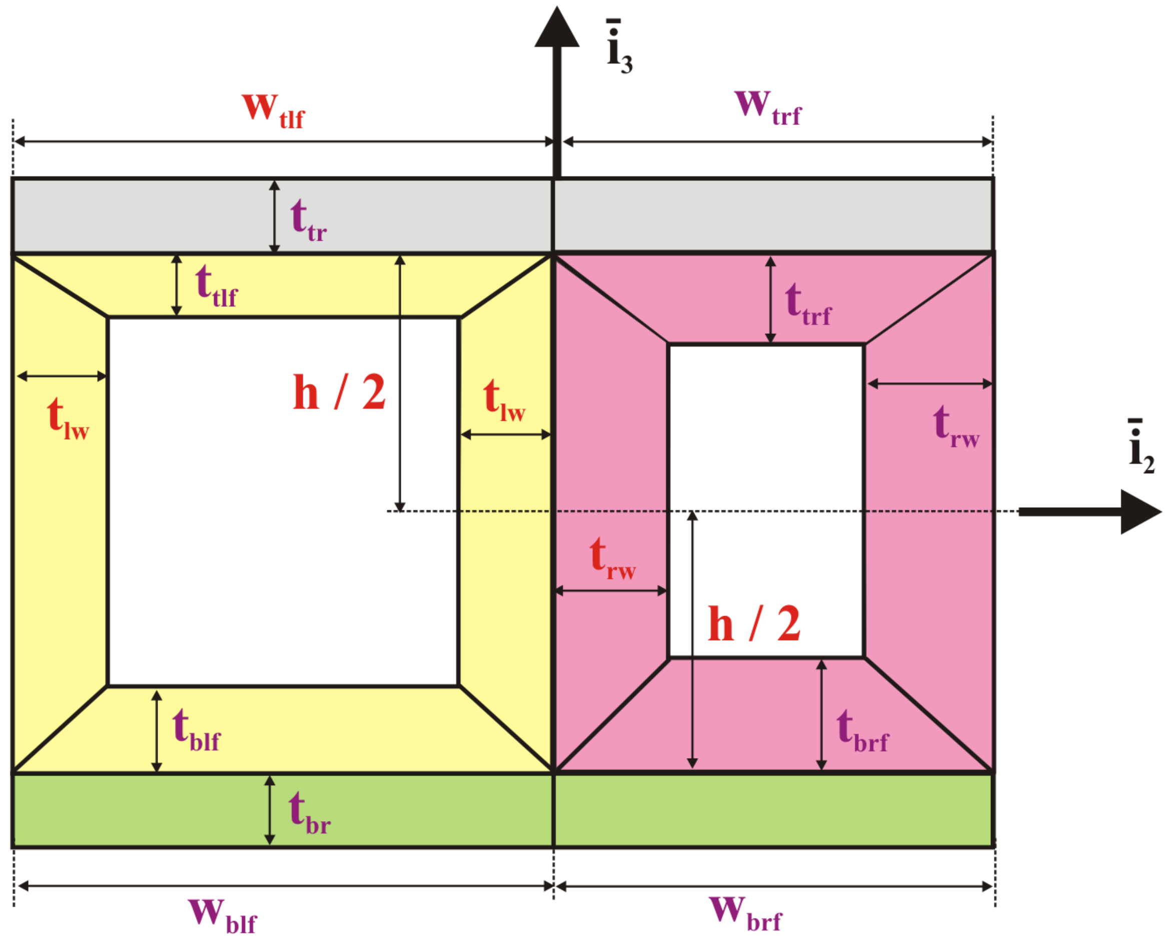

Figure 2.39: Configuration of the double box.

Double boxes are defined by means of three dialog tabs.

1. The *Double Box* dialog tab as described in section 2.5.1, which defines the name of the section.
2. The *Dimensions* dialog tab as described in section 2.5.2, which defines the dimensions of the section.
3. The *Materials* dialog tab as described in section 2.5.3, which defines the materials the section is made of.

2.5.1 The *Double Box* dialog tab

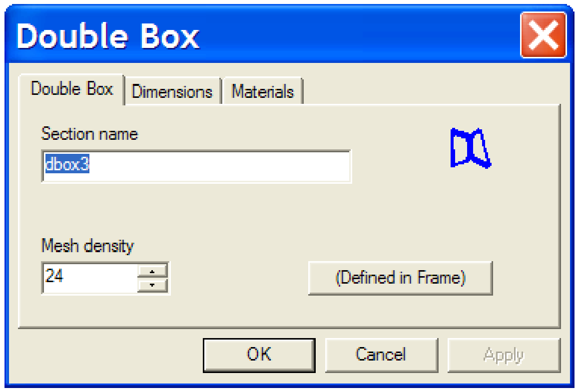

Figure 2.40: The *Double Box* dialog tab.

The *Double Box* dialog tab as described in fig. 2.40, defines the following data for the double box.

1. **Section name.** Enter a unique name for the double box.

2. **Mesh density.** Enter the desired mesh density as described in section 5.1, for the finite element discretization.

3. *(Defined in frame).* The geometry of the double box can be defined with respect to a fixed frame as described in section 7.1, allowing translation and rotation of the section as a rigid body.

2.5.2 The *Dimensions* dialog tab

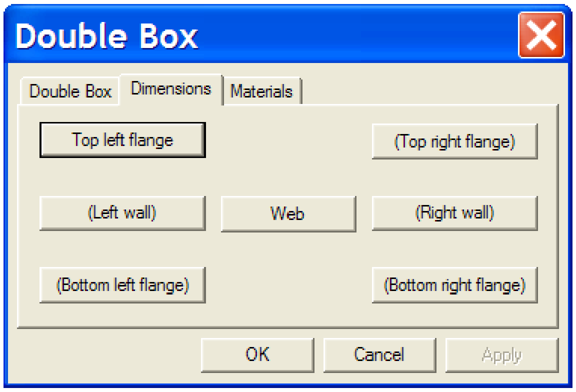

Figure 2.41: The *Dimensions* dialog tab.

The *Dimensions* dialog tab as described in fig. 2.5.2, defines the dimensions of the double box shown in fig. 2.39. The dimensions of the section are defined by the following parameters.

Web dimensions

1. The height, h, of the section (**required input**).
2. The thickness, t_{lw}, of the left part of the web (**required input**).
3. The thickness, t_{rw}, of the right part of the web (**required input**).

Top left flange dimensions

1. The width, w_{tlf}, of the top left flange (**required input**).
2. The thickness, t_{tlf}, of the top left flange (*default value:* $t_{\mathrm{tlf}} = t_{\mathrm{lw}}$).
3. The skew angle, α_{tlf}, of the top left flange, positive up, measured in degrees (*default value:* $\alpha_{\mathrm{tlf}} = 0$).
4. The thickness, t_{tr}, of the top reinforcement flange; this thickness applies to both left and right reinforcements that cannot exist independently of each other (*default value:* $t_{\mathrm{tr}} = 0$). This variable is also used as a flag for the presence of the top flange reinforcement: if $t_{\mathrm{tr}} > 0$, this reinforcement is present.

Top right flange dimensions

1. The width, w_{trf}, of the top right flange (*default value:* $w_{\mathrm{trf}} = w_{\mathrm{tlf}}$).
2. The thickness, t_{trf}, of the top right flange (*default value:* $t_{\mathrm{trf}} = t_{\mathrm{rw}}$).
3. The skew angle, α_{trf}, of the top right flange, positive up, measured in degrees (*default value:* $\alpha_{\mathrm{trf}} = 0$).

Bottom left flange dimensions

1. The width, w_{blf}, of the bottom left flange (*default value:* $w_{\mathrm{blf}} = w_{\mathrm{tlf}}$).
2. The thickness, t_{blf}, of the bottom left flange (*default value:* $t_{\mathrm{blf}} = t_{\mathrm{tlf}}$).
3. The skew angle, α_{blf}, of the bottom left flange, positive down, measured in degrees (*default value:* $\alpha_{\mathrm{blf}} = 0$).

4. The thickness, t_{br}, of the bottom reinforcement flange; this thickness applied to both left and right reinforcements that cannot exist independently of each other (*default value:* $t_{\mathrm{br}} = 0$). This variable is also used as a flag for the presence of the bottom flange reinforcement: if $t_{\mathrm{br}} > 0$, this reinforcement is present.

Bottom right flange dimensions

1. The width, w_{brf}, of the bottom right flange (*default value:* $w_{\mathrm{brf}} = w_{\mathrm{trf}}$).
2. The thickness, t_{brf}, of the bottom right flange (*default value:* $t_{\mathrm{brf}} = t_{\mathrm{trf}}$).
3. The skew angle, α_{brf}, of the bottom right flange, positive down, measured in degrees (*default value:* $\alpha_{\mathrm{brf}} = 0$).

2.5.3 The *Materials* dialog tab

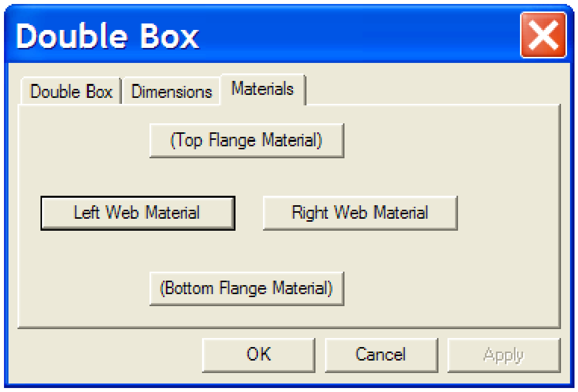

Figure 2.42: The *Materials* dialog tab.

The *Materials* dialog tab as described in fig. 2.42, defines the materials the double box is made of. As shown in fig. 2.43, the section is divided into four zones.

1. The top reinforcement flange consists of components labeled *TLFlgR* and *TRFlgR*
2. The left portion of the double box itself consists of components labeled **TLFlg**, **LWeb** and **BLFlg**.
3. The right portion of the double box itself consists of components labeled **TRFlg**, **RWeb** and **BRFlg**.
4. The top reinforcement flange consists of components labeled *BLFlgR* and *BRFlgR*.

It is possible to assign material properties as described in section 4.1, or solid properties as described in section 4.2, to each zone of the double box.

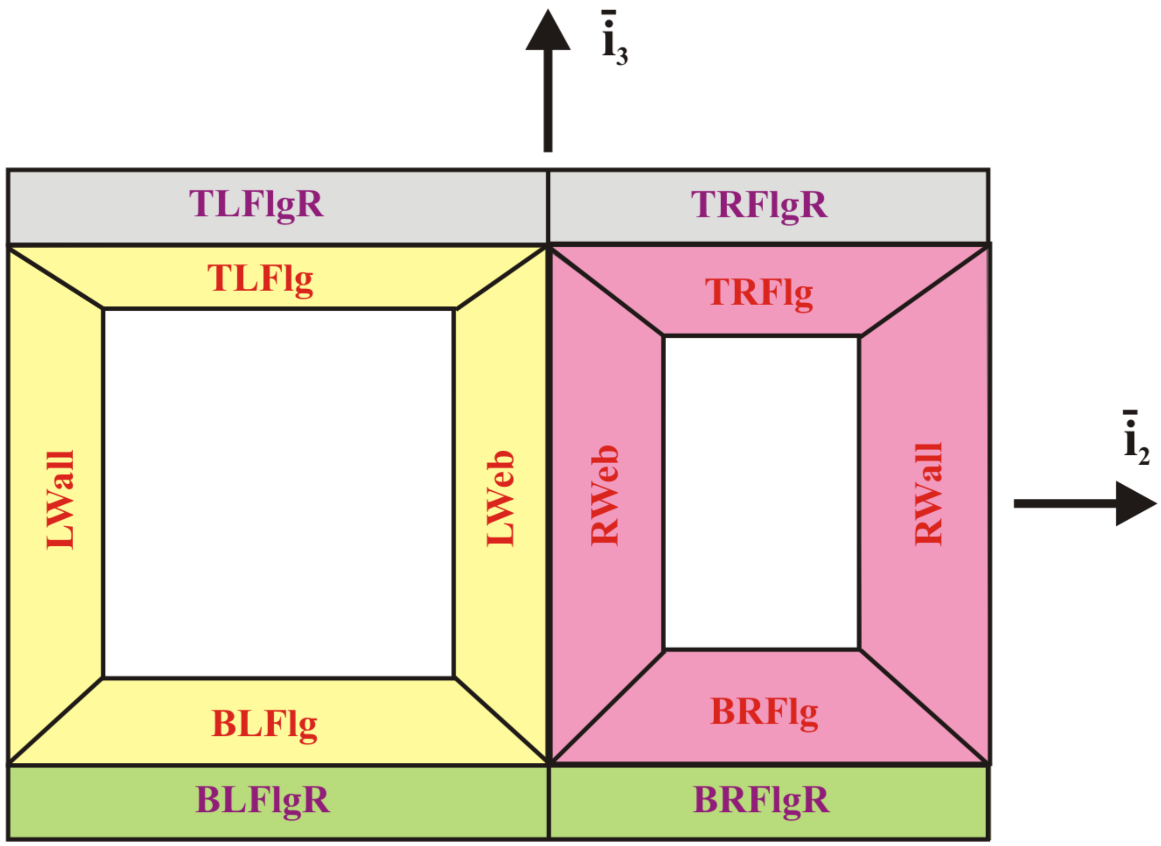

Figure 2.43: The four zones of the double box.

2.5.4 Formatted input

The data defined in the above sections will be saved in a specially formatted input file which has the following structure.

@DOUBLE_BOX_DEFINITION {

@DOUBLE_BOX_NAME { `DboxName` **} {**

@WEB_HEIGHT { h **}**
@LEFT_WEB_THICKNESS { t_{lw} **}**
@RIGHT_WEB_THICKNESS { t_{rw} **}**
@TOP_FLANGE_WIDTH { w_{tf} **}**
@TOP_FLANGE_THICKNESS { t_{tf} }
@TOP_FLANGE_SKEW_ANGLE { α_{tf} }
@TOP_FLANGE_REINFORCE_THICKNESS { t_{tr} }
@BOTTOM_FLANGE_WIDTH { w_{bf} }
@BOTTOM_FLANGE_THICKNESS { t_{bf} }
@BOTTOM_FLANGE_SKEW_ANGLE { α_{bf} }
@BOTTOM_FLANGE_REINFORCE_THICKNESS { t_{br} }
@LEFT_WEB_MATERIAL_NAME { `LWebMaterialName` }
@RIGHT_WEB_MATERIAL_NAME { `RWebMaterialName` }
@TOP_REINFORCE_MATERIAL_NAME { `TopMaterialName` }
@BOTTOM_REINFORCE_MATERIAL_NAME { `BottomMaterialName` }
@IS_DEFINED_IN_FRAME { `FxdFrameName` }
@MESH_DENSITY { md }

}

}

2.5.5 Examples

A few examples that describe the construction procedure of this type of section are shown below.

Example 1
This example shows a double box. Here web height, web thicknesses, flange width, web materials and mesh density are assigned for constructing this section. This example also shows the axial strain field over the cross-section under the applied sectional loads.

Example 2
This example shows a double box with skewed flanges. Here web height, web thicknesses, flange widths,flange thicknesses, flange reinforcement thickness, flange skew angles, materials and mesh density are assigned for constructing this section. This example also shows the axial stress field over the cross-section under the applied sectional loads.

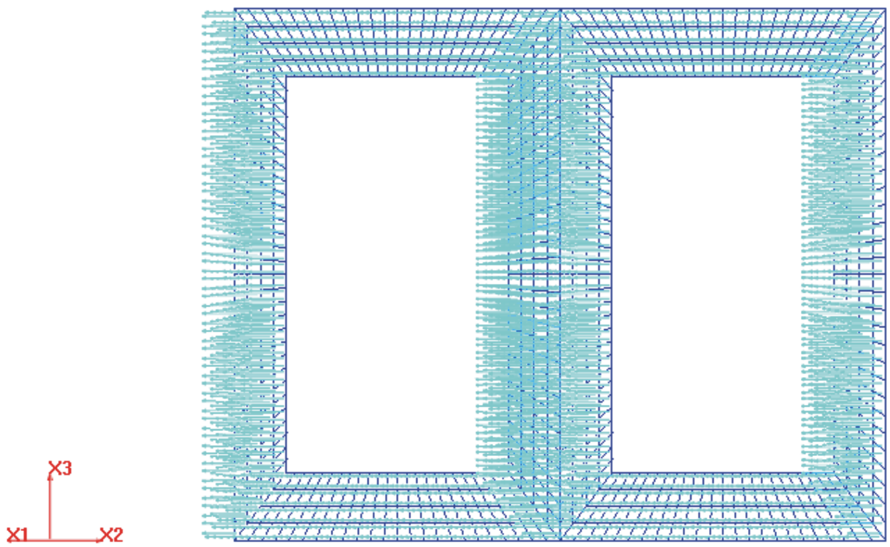

Figure 2.44: Example 1-Double Box.

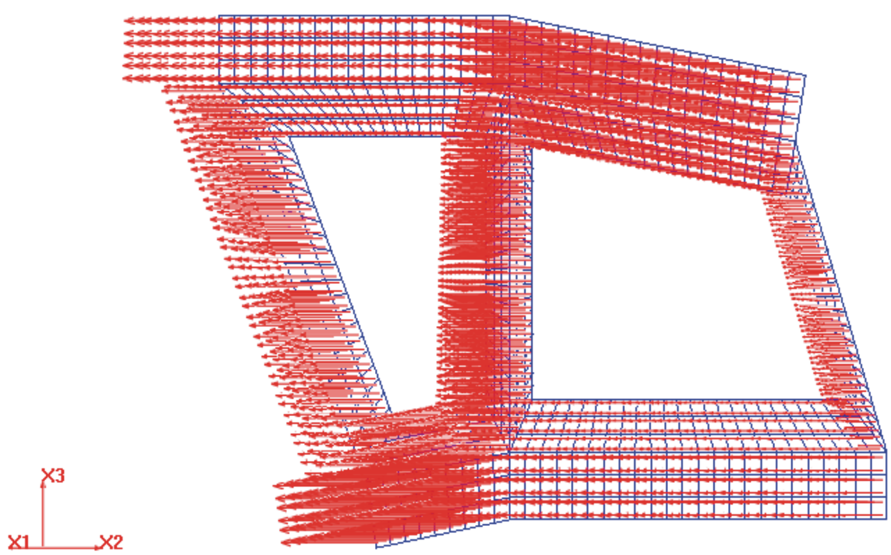

Figure 2.45: Example 2-Double Box.

2.6 Definition of I-sections

I-sections are parametric configurations of the shape depicted in fig. 2.46. They consist of an I-section, possibly reinforced by top and/or bottom flanges. The section consists of up to four zones to which independent material properties can be assigned. Note that through the use of fixed frames as described in section 7.1, the I-section can be made to look like the following shapes: H-sections.

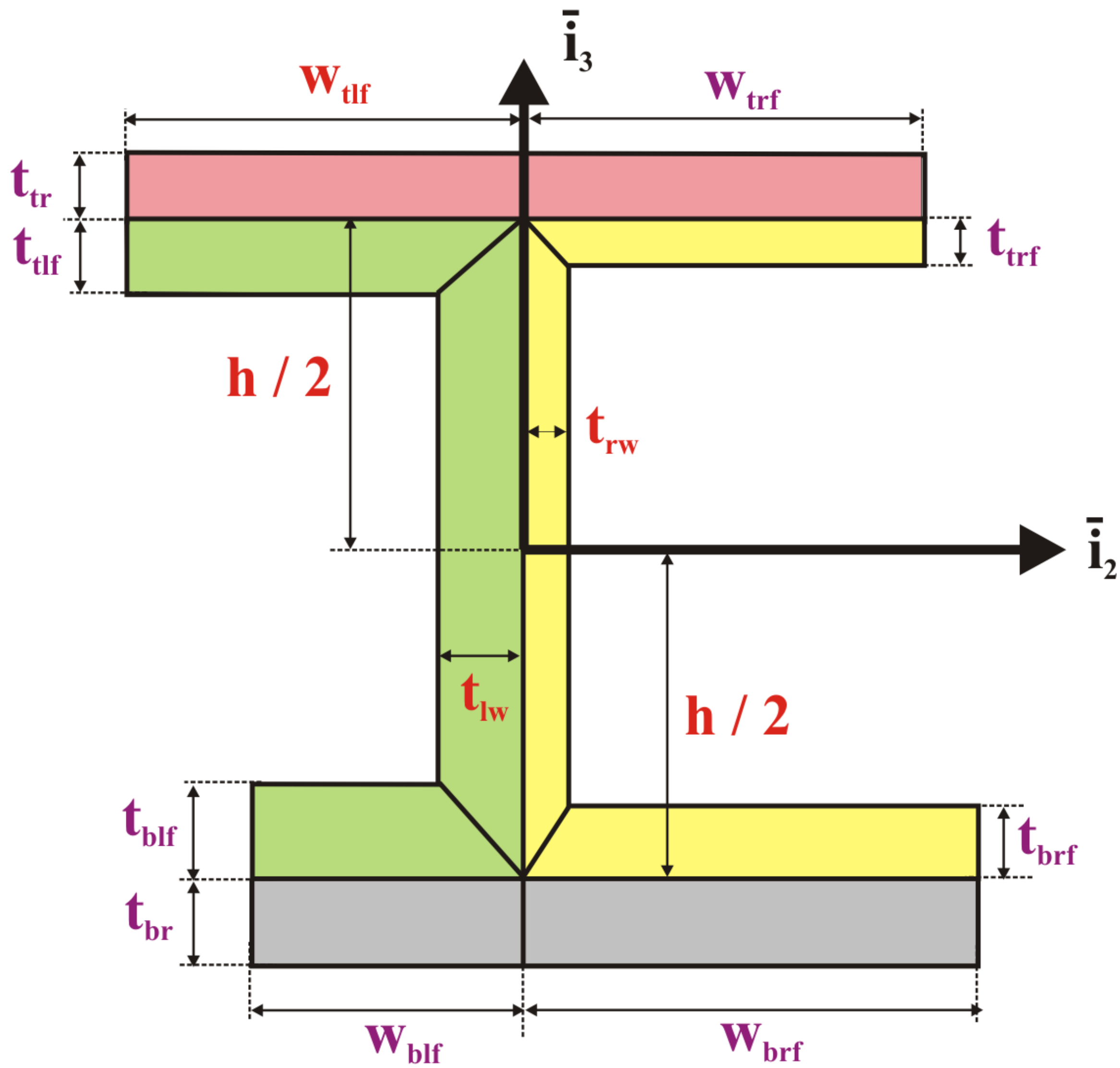

Figure 2.46: Configuration of the I-section.

I-sections are defined by means of three dialog tabs.

1. The *I-section* dialog tab as described in section 2.6.1, which defines the name of the section.
2. The *Dimensions* dialog tab as described in section 2.6.2, which defines the dimensions of the section.
3. The *Materials* dialog tab as described in section 2.6.3, which defines the materials the section is made of.

2.6.1 The *I-section* dialog tab

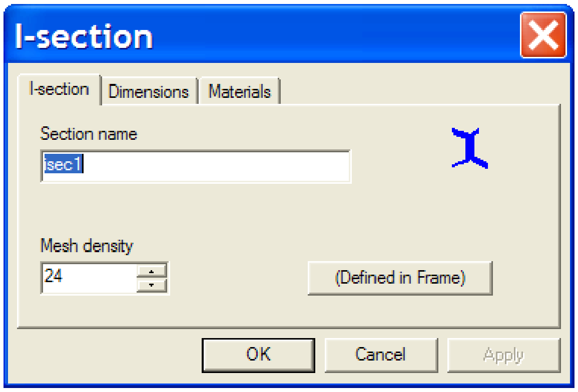

Figure 2.47: The *I-section* dialog tab.

The *I-section* dialog tab as described in fig. 2.47, defines the following data for the I-section.

1. **Section name.** Enter a unique name for the I-section.
2. **Mesh density.** Enter the desired mesh density as described in section 5.1, for the finite element discretization.
3. *(Defined in frame).* The geometry of the I-section can be defined with respect to a fixed frame as described in section 7.1, allowing translation and rotation of the section as a rigid body.

2.6.2 The *Dimensions* dialog tab

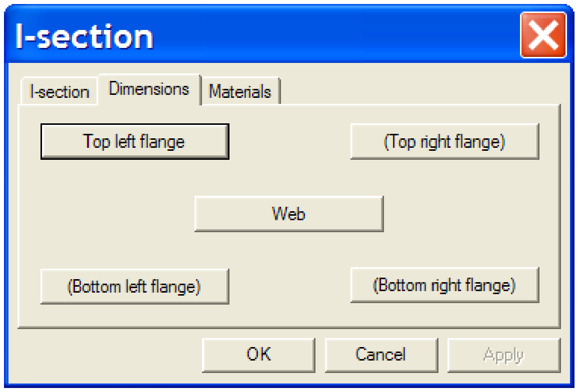

Figure 2.48: The *Dimensions* dialog tab.

The *Dimensions* dialog tab as described in fig. 2.6.2, defines the dimensions of the I-section shown in fig. 2.46. The dimensions of the section are defined by the following parameters.

Web dimensions

1. The height, h, of the section (**required input**).
2. The thickness, t_{lw}, of the left part of the web (**required input**).
3. The thickness, t_{rw}, of the right part of the web (**required input**).

Top left flange dimensions

1. The width, w_{tlf}, of the top left flange (**required input**).
2. The thickness, t_{tlf}, of the top left flange (*default value:* $t_{\mathrm{tlf}} = t_{\mathrm{lw}}$).
3. The skew angle, α_{tlf}, of the top left flange, positive up, measured in degrees (*default value:* $\alpha_{\mathrm{tlf}} = 0$).
4. The thickness, t_{tr}, of the top reinforcement flange; this thickness applies to both left and right reinforcements that cannot exist independently of each other (*default value:* $t_{\mathrm{tr}} = 0$). This variable is also used as a flag for the presence of the top flange reinforcement: if $t_{\mathrm{tr}} > 0$, this reinforcement is present.

Top right flange dimensions

1. The width, w_{trf}, of the top right flange (*default value:* $w_{\mathrm{trf}} = w_{\mathrm{tlf}}$).
2. The thickness, t_{trf}, of the top right flange (*default value:* $t_{\mathrm{trf}} = t_{\mathrm{rw}}$).
3. The skew angle, α_{trf}, of the top right flange, positive up, measured in degrees (*default value:* $\alpha_{\mathrm{trf}} = 0$).

Bottom left flange dimensions

1. The width, w_{blf}, of the bottom left flange (*default value:* $w_{\mathrm{blf}} = w_{\mathrm{tlf}}$).
2. The thickness, t_{blf}, of the bottom left flange (*default value:* $t_{\mathrm{blf}} = t_{\mathrm{tlf}}$).
3. The skew angle, α_{blf}, of the bottom left flange, positive down, measured in degrees (*default value:* $\alpha_{\mathrm{blf}} = 0$).

4. The thickness, t_{br}, of the bottom reinforcement flange; this thickness applied to both left and right reinforcements that cannot exist independently of each other (*default value:* $t_{\mathrm{br}} = 0$). This variable is also used as a flag for the presence of the bottom flange reinforcement: if $t_{\mathrm{br}} > 0$, this reinforcement is present.

Bottom right flange dimensions

1. The width, w_{brf}, of the bottom right flange (*default value:* $w_{\mathrm{brf}} = w_{\mathrm{trf}}$).
2. The thickness, t_{brf}, of the bottom right flange (*default value:* $t_{\mathrm{brf}} = t_{\mathrm{trf}}$).
3. The skew angle, α_{brf}, of the bottom right flange, positive down, measured in degrees (*default value:* $\alpha_{\mathrm{brf}} = 0$).

2.6.3 The *Materials* dialog tab

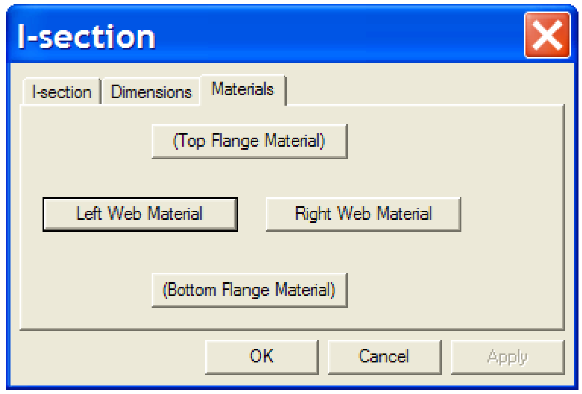

Figure 2.49: The *Materials* dialog tab.

The *Materials* dialog tab as described in fig. 2.49, defines the materials the I-section is made of. As shown in fig. 2.50, the section is divided into four zones.

1. The top reinforcement flange consists of components labeled *TLFlgR* and *TRFlgR*
2. The left portion of the I-section itself consists of components labeled **TLFlg**, **LWeb** and **BLFlg**.
3. The right portion of the I-section itself consists of components labeled **TRFlg**, **RWeb** and **BRFlg**.
4. The top reinforcement flange consists of components labeled *BLFlgR* and *BRFlgR*.

It is possible to assign material properties as described in section 4.1, or solid properties as described in section 4.2, to each zone of the I-section.

Special Cases

1. If the widths, $w_{\mathrm{trf}} = 0$ and $w_{\mathrm{blf}} = 0$, the section looks like Z-shape depicted in fig. 2.51. This Z-section does not have TRFlgR, TRFlg, BLFlgR and BLFlg solids shown in fig. 2.52. Here the coordinates of the points are also adjusted from the I-section.
2. If the widths, $w_{\mathrm{blf}} = 0$ and $w_{\mathrm{brf}} = 0$, the section looks like T-shape depicted in fig. 2.53. This T-section does not have BLFlgR, BLFlg, BRFlgR, and BRFlg solids shown in fig. 2.54. Here the coordinates of the points are also adjusted from the I-section.

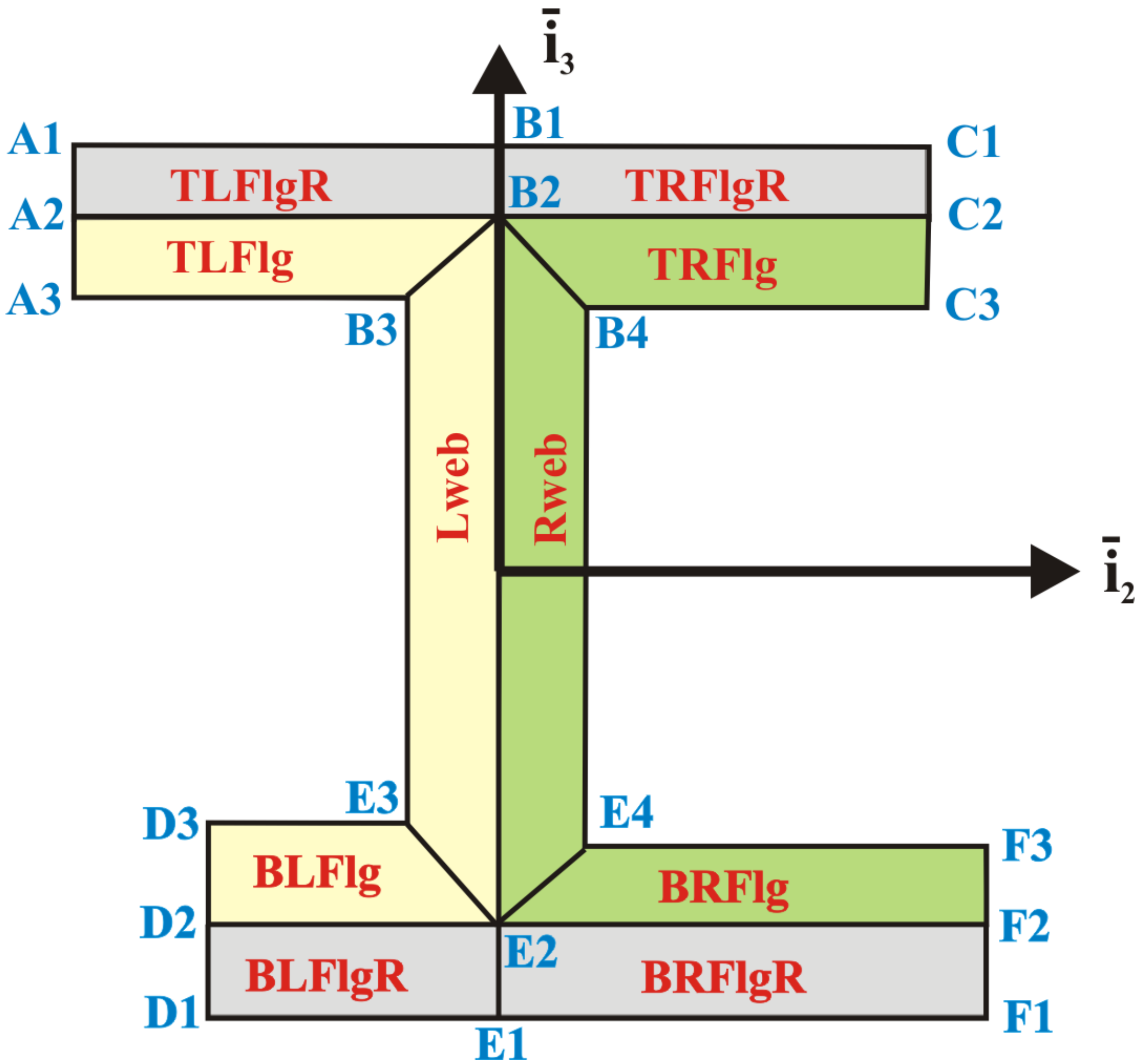

Figure 2.50: The four zones of the I-Section.

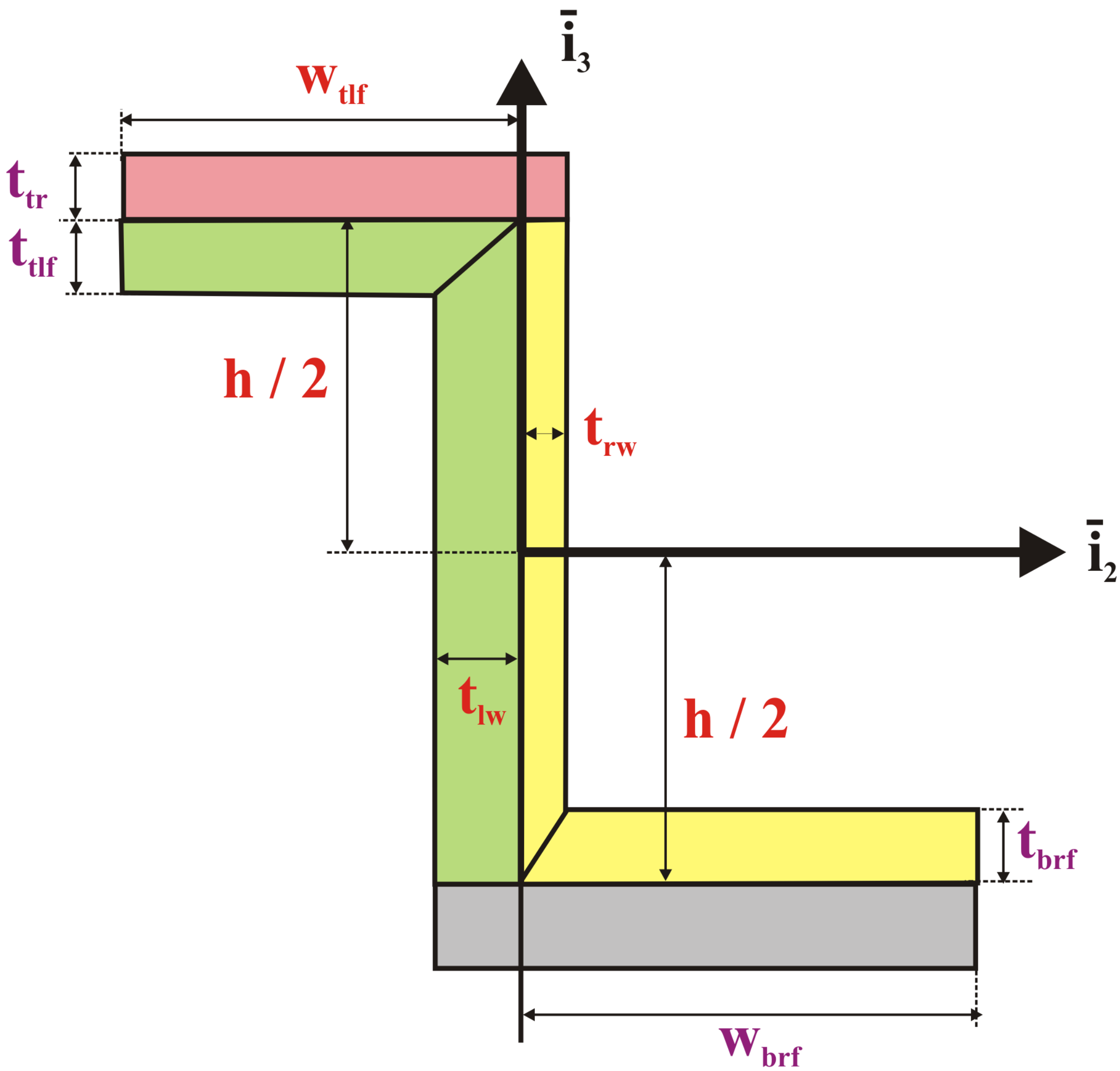

Figure 2.51: Configuration of the Z-section.

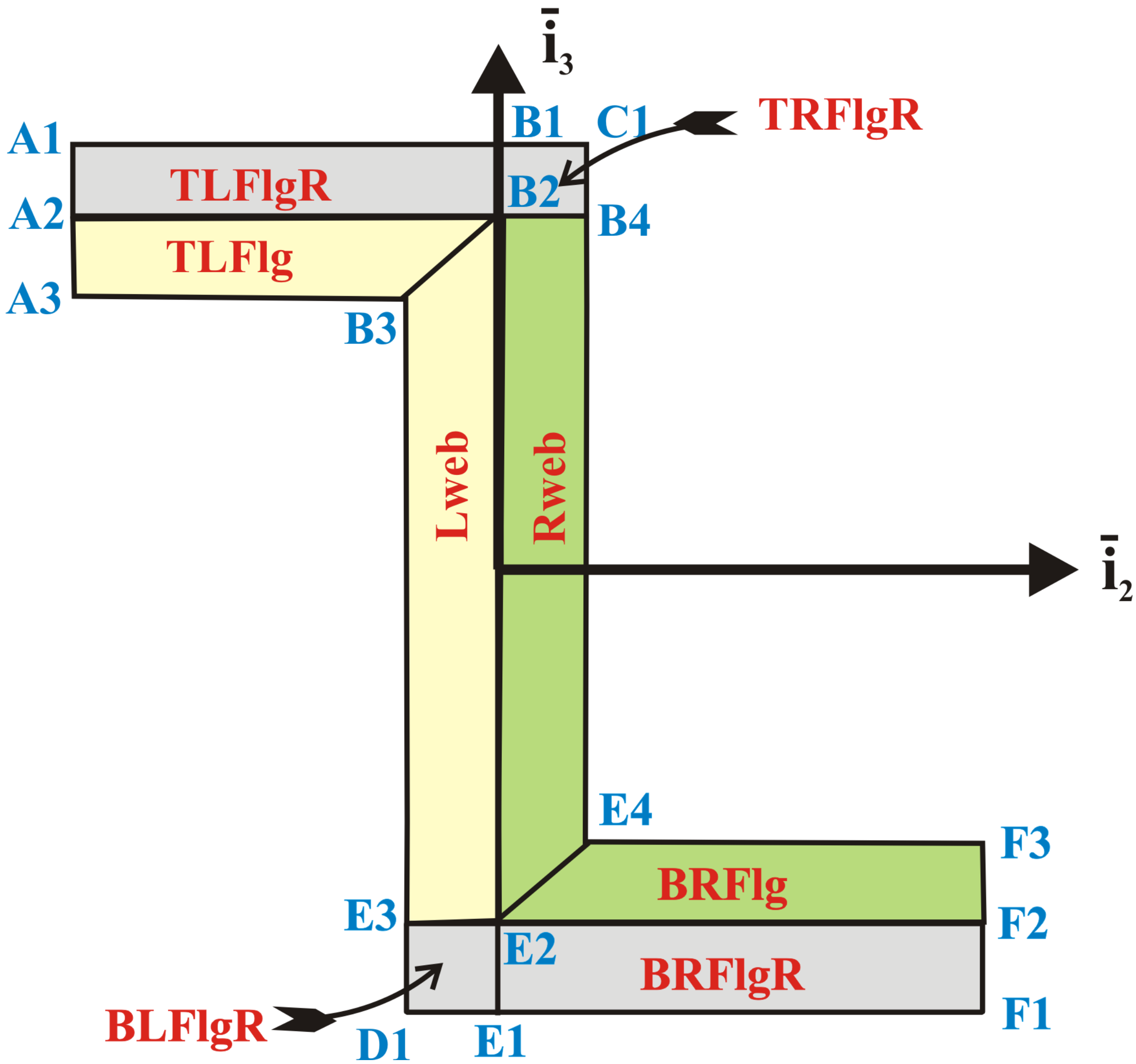

Figure 2.52: The zones of the Z-section.

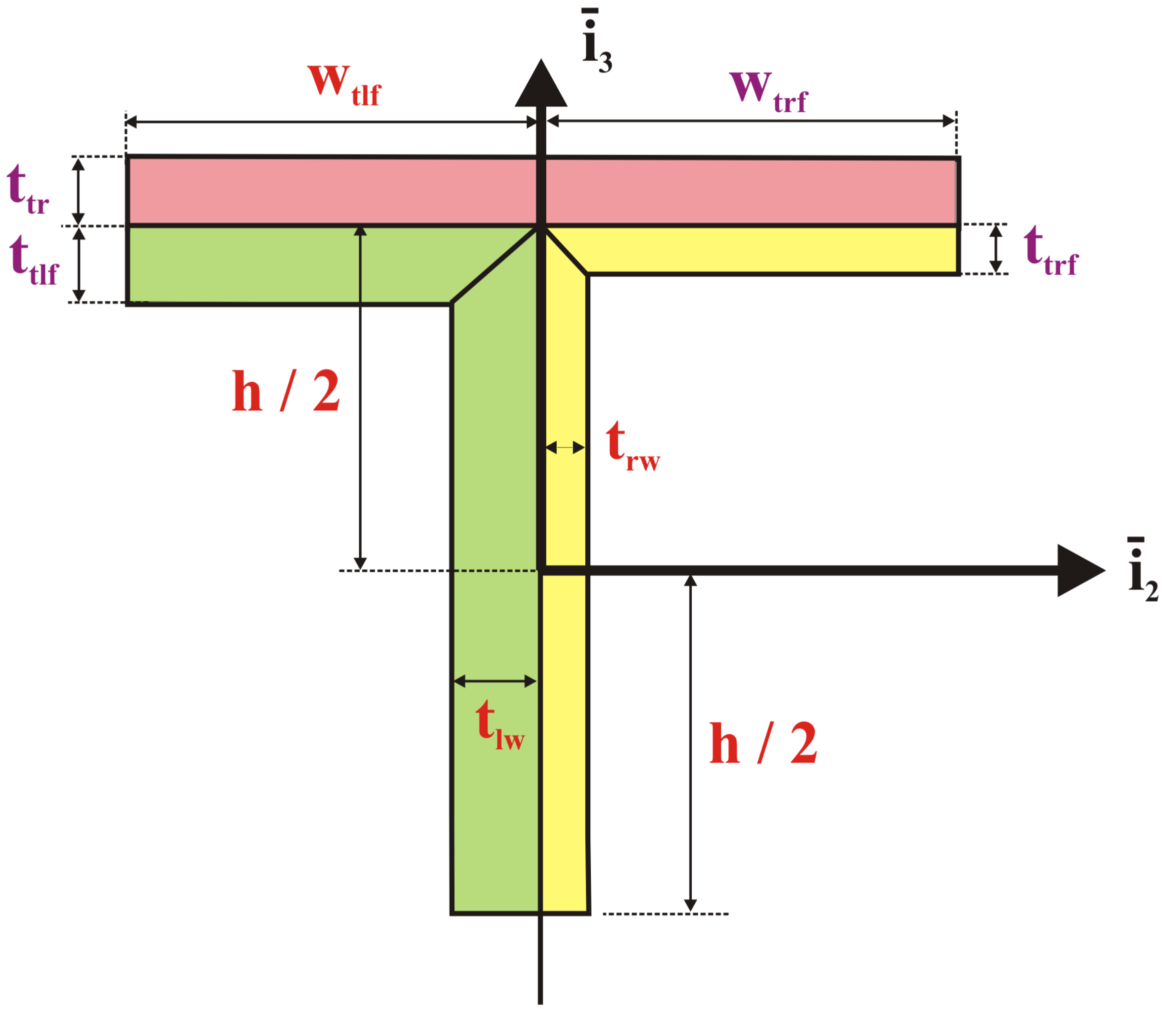

Figure 2.53: Configuration of the T-section.

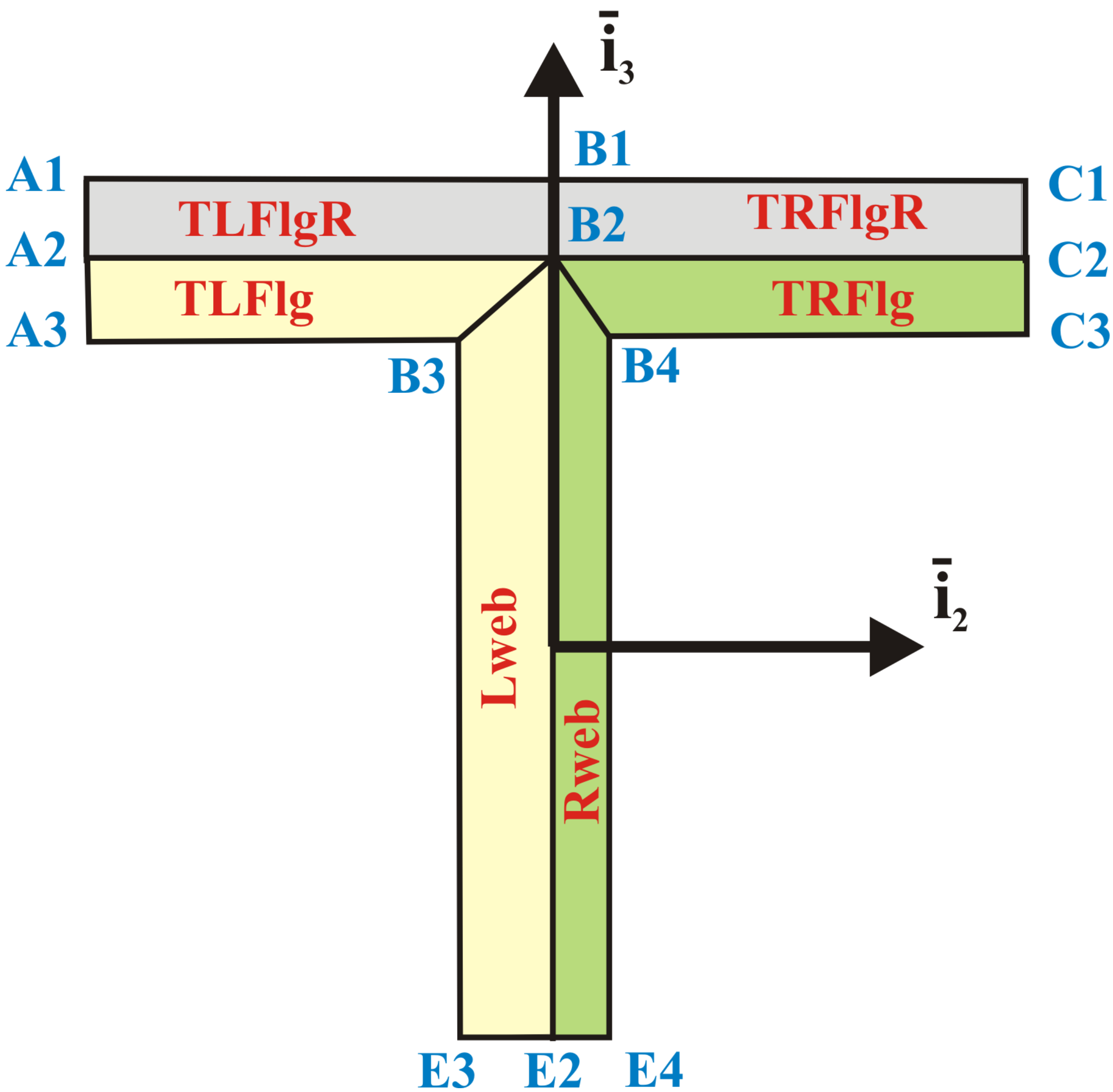

Figure 2.54: The zones of the T-section.

2.6.4 Formatted input

The data defined in the above sections will be saved in a specially formatted input file which has the following structure.

@I_SECTION_DEFINITION {

@I_SECTION_NAME { `IsecName` **} {**

@WEB_HEIGHT { h **}**
@LEFT_WEB_THICKNESS { t_{lw} **}**
@RIGHT_WEB_THICKNESS { t_{rw} **}**
@TOP_FLANGE_WIDTH { w_{tf} **}**
@TOP_FLANGE_THICKNESS { t_{tf} }
@TOP_FLANGE_SKEW_ANGLE { α_{tf} }
@TOP_FLANGE_REINFORCE_THICKNESS { t_{tr} }
@BOTTOM_FLANGE_WIDTH { w_{bf} }
@BOTTOM_FLANGE_THICKNESS { t_{bf} }
@BOTTOM_FLANGE_SKEW_ANGLE { α_{bf} }
@BOTTOM_FLANGE_REINFORCE_THICKNESS { t_{br} }
@LEFT_WEB_MATERIAL_NAME { `LWebMaterialName` }
@RIGHT_WEB_MATERIAL_NAME { `RWebMaterialName` }
@TOP_REINFORCE_MATERIAL_NAME { `TopMaterialName` }
@BOTTOM_REINFORCE_MATERIAL_NAME { `BottomMaterialName` }
@IS_DEFINED_IN_FRAME { `FxdFrameName` }
@MESH_DENSITY { md }

}

}

2.6.5 Examples

A few examples that describe the construction procedure of this type of section are shown below.

Example 1
This example shows an I-section. Here web height, web thicknesses, flange widths, flange thicknesses, web materials and mesh density are assigned for constructing this section. This example also shows the shear stress field over the cross-section under the applied sectional loads.
Example 2
This example shows an I-section with top and bottom reinforcements. Here web height, web thicknesses, flange widths, flange reinforcement thicknesses, materials and mesh density are assigned for constructing this section. This example also shows the axial strain field over the cross-section under the applied sectional loads.
Example 3
This example also shows a Z-section without any reinforcement. Here web height, web thicknesses, flange widths (here $w_{\text{trf}} = 0$ and $w_{\text{blf}} = 0$), materials and mesh density are assigned for constructing this section. This example also shows principal centroidal axes of bending.
Example 4
This example shows a Z-section with top and bottom reinforcements. Here web height, web thicknesses, flange widths (here $w_{\text{trf}} = 0$ and $w_{\text{blf}} = 0$), flange reinforcement thicknesses, materials and mesh density are

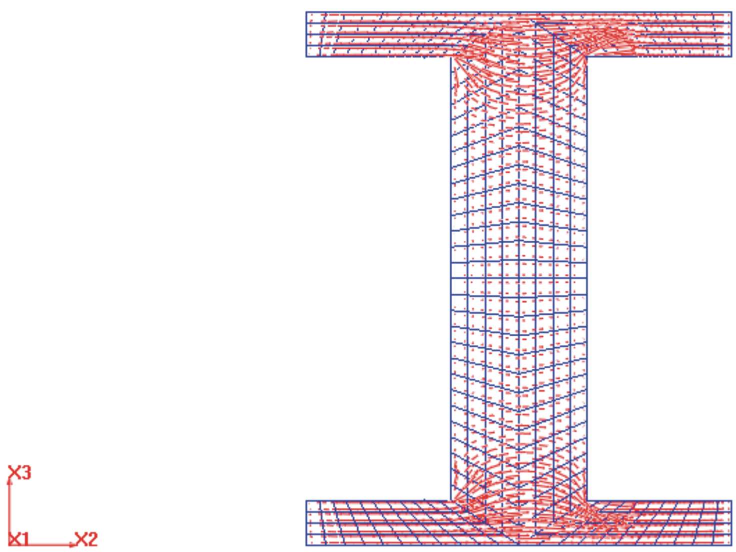

Figure 2.55: Example 1.I-section

assigned for constructing this section. This example also shows the axial stress field over the cross-section under the applied sectional loads.

Example 5

This example shows a T-section with top reinforcement. Here web height, web thicknesses, flange widths (here $w_{\text{blf}} = 0$ and $w_{\text{brf}} = 0$), flange reinforcement thicknesses, materials and mesh density are assigned for constructing this section. This example also shows the principal axes of inertia at the mass center.

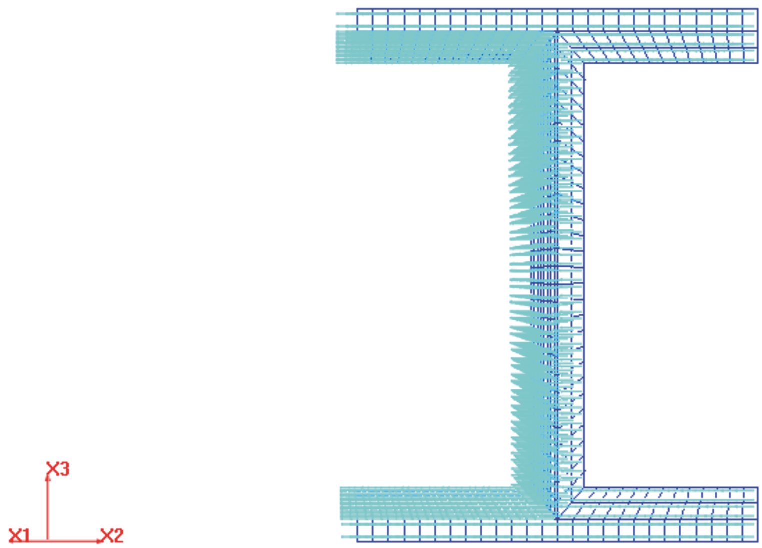

Figure 2.56: Example 2.I-section

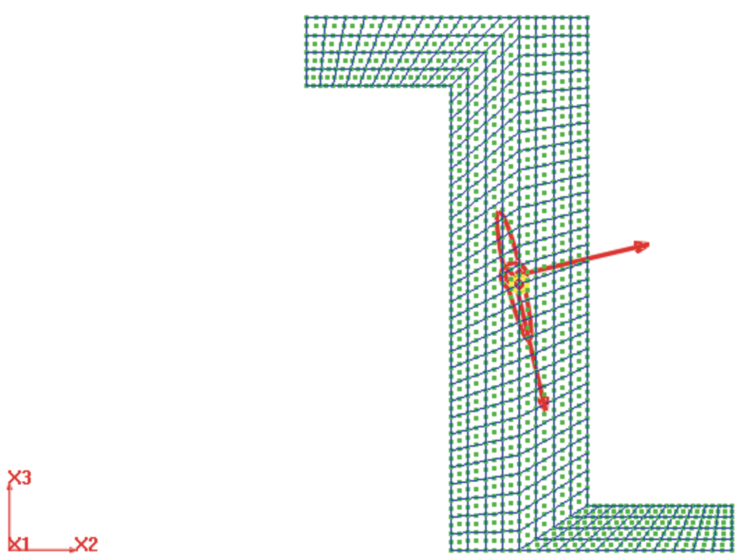

Figure 2.57: Example 3.Z-section

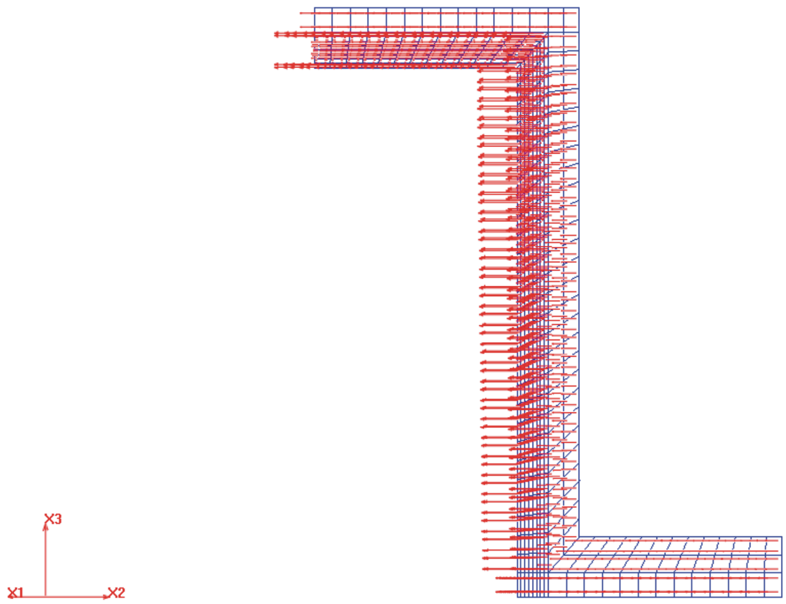

Figure 2.58: Example 4.Z-section

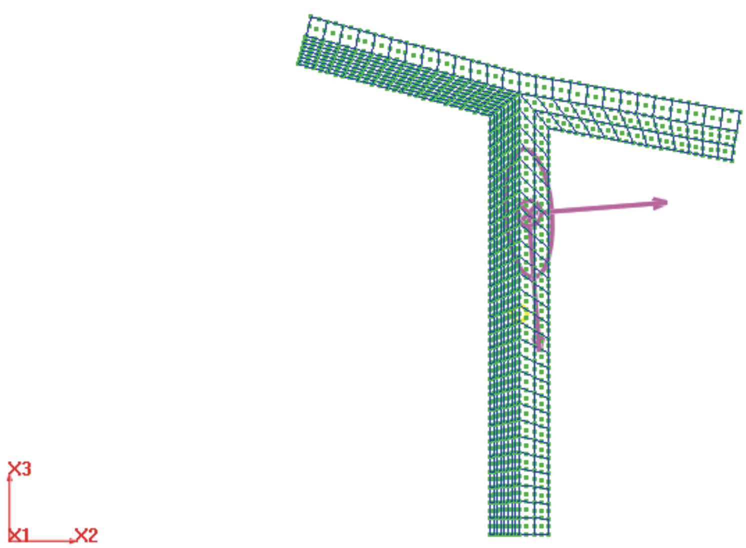

Figure 2.59: Example 5.T-section

2.7 Definition of rectangular boxes

Rectangular boxes are parametric configurations of the shape depicted in fig. 2.60. They consist of a rectangular box, possibly reinforced by top and/or bottom flanges. The section consists of up to three three zones to which independent material properties can be assigned.

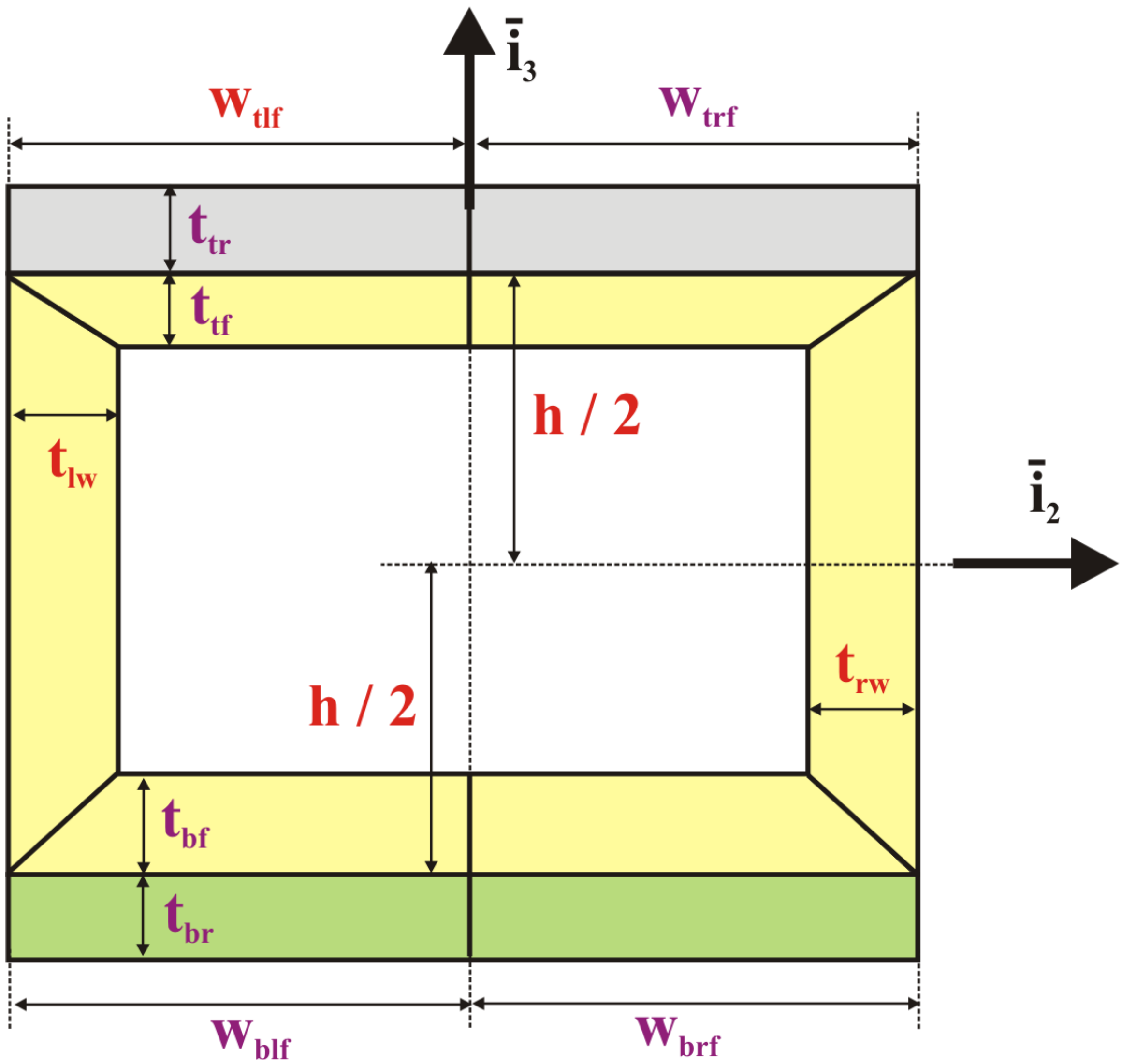

Figure 2.60: Configuration of the rectangular box. The dimensions of the various elements of the section are indicated on the figure. The three shaded areas correspond to the three zones of three section.

Rectangular boxes are defined by means of three dialog tabs.

1. The *Rectangular box* dialog tab as described in section 2.7.1, which defines the name of the section.
2. The *Dimensions* dialog tab as described in section 2.7.2, which defines the dimensions of the section.
3. The *Materials* dialog tab as described in section 2.7.3, which defines the materials the section is made of.

2.7.1 The *Rectangular box* dialog tab

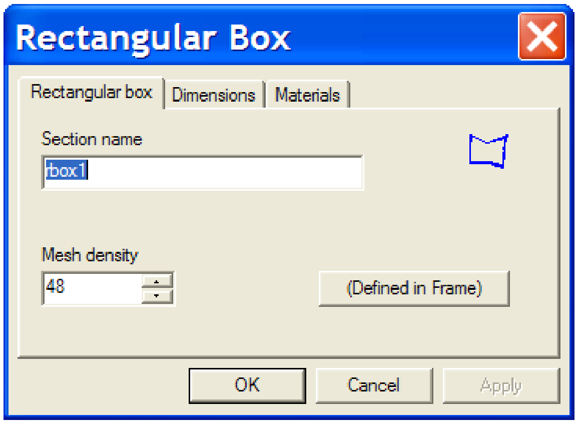

Figure 2.61: The *Rectangular box* dialog tab.

The *Rectangular box* dialog tab as described in fig. 2.61, defines the following data for the rectangular box.

1. **Section name.** Enter a unique name for the rectangular box.
2. **Mesh density.** Enter the desired mesh density as described in section 5.1, for the finite element discretization.
3. *(Defined in frame).* The geometry of the rectangular box can be defined with respect to a fixed frame as described in section 7.1, allowing translation and rotation of the section as a rigid body.

2.7.2 The *Dimensions* dialog tab

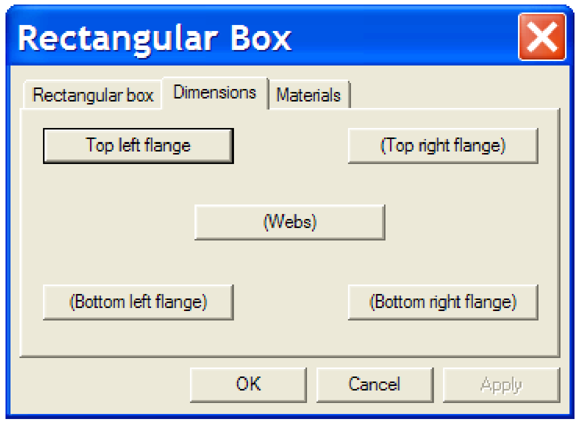

Figure 2.62: The *Dimensions* dialog tab.

The *Dimensions* dialog tab as described in fig. 2.62, defines the dimensions of the rectangular box, as depicted in fig. 2.60. Dimensions of the section are defined by the following parameters.

Web dimensions

1. The height, h, of the section (**required input**).
2. The thickness, t_{lw}, of the left web (**required input**).
3. The thickness, t_{rw}, of the right web (**required input**).

Top left flange dimensions

1. The width, w_{tlf}, of the top left flange (**required input**).
2. The thickness, t_{tf}, of the top flange (*default value:* $t_{\mathrm{tf}} = t_{\mathrm{lw}}$).
3. The skew angle, α_{tlf}, of the top left flange, positive up, measured in degrees (*default value:* $\alpha_{\mathrm{tlf}} = 0$).
4. The thickness, t_{tr}, of the top reinforcement flange; this thickness applies to both left and right reinforcements that cannot exist independently of each other (*default value:* $t_{\mathrm{tr}} = 0$). This variable is also used as a flag for the presence of the top reinforcement flange: if $t_{\mathrm{tr}} \neq 0$, the top reinforcement flange is present.

Top right flange dimensions

1. The width, w_{trf}, of the top right flange (*default value:* $w_{\mathrm{trf}} = w_{\mathrm{tlf}}$).
2. The thickness of the top right flange equals that of the top left flange.
3. The skew angle, α_{trf}, of the top right flange, positive up, measured in degrees (*default value:* $\alpha_{\mathrm{trf}} = 0$).

Bottom left flange dimensions

1. The width, w_{blf}, of the bottom left flange (*default value:* $w_{\mathrm{blf}} = w_{\mathrm{tlf}}$).
2. The thickness, t_{bf}, of the bottom left flange (*default value:* $t_{\mathrm{bf}} = t_{\mathrm{tf}}$).

3. The skew angle, α_{blf}, of the bottom left flange, positive down, measured in degrees (*default value:* $\alpha_{\mathrm{blf}} = 0$).

4. The thickness, t_{br}, of the bottom reinforcement flange; this thickness applied to both left and right reinforcements that cannot exist independently of each other (*default value:* $t_{\mathrm{br}} = 0$). This variable is also used as a flag for the presence of the bottom reinforcement flange: if $t_{\mathrm{br}} \neq 0$, the bottom reinforcement flange is present.

Bottom right flange dimensions

1. The width, w_{brf}, of the bottom right flange (*default value:* $w_{\mathrm{brf}} = w_{\mathrm{blf}}$).

2. The thickness of the bottom right flange equals that of the bottom left flange.

3. The skew angle, α_{brf}, of the bottom right flange, positive down, measured in degrees (*default value:* $\alpha_{\mathrm{brf}} = 0$).

2.7.3 The *Materials* dialog tab

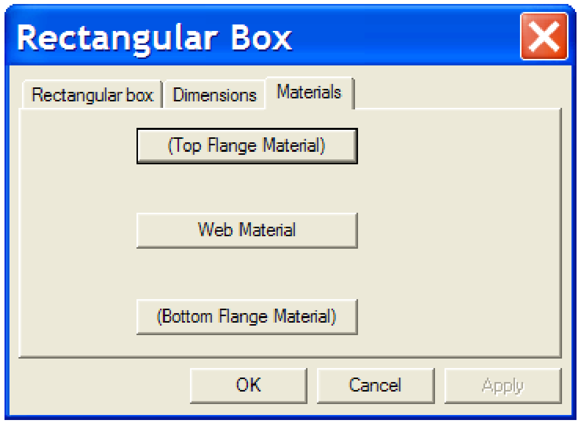

Figure 2.63: The *Materials* dialog tab.

The *Materials* dialog tab as described in fig. 2.63, defines the materials the rectangular box is made of. As shown in fig. 2.64, the section is divided into three zones.

1. The top reinforcement flange consists of the components labeled *TLFlgR* and *TRFlgR*.

2. The rectangular box consists of the components labeled **TLFlg**, **TRFlg**, **LWeb**, **RWeb**, **BLFlg**, and **BRFlg**.

3. The bottom reinforcement flange consists of the components labeled *BLFlgR* and *BRFlgR*.

It is possible to assign material properties as described in section 4.1, or solid properties as described in section 4.2, to each zone of the rectangular box.

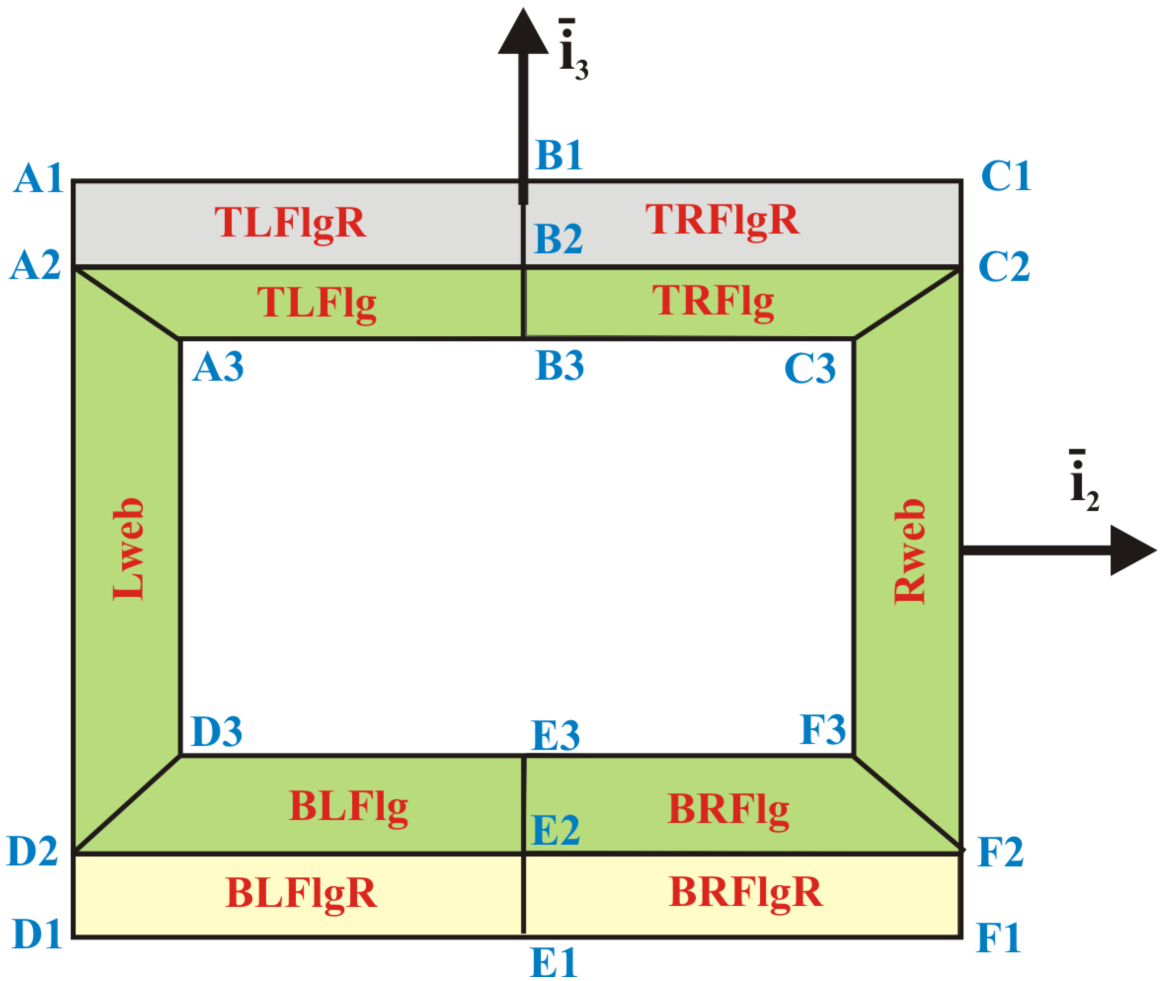

Figure 2.64: The three zones of the rectangular box.

2.7.4 Formatted input

The data that defines the rectangular box as described in the above sections will be saved in a specially formatted input file which has the following structure.

@RECTANGULAR_BOX_DEFINITION {

@RECTANGULAR_BOX_NAME { `RBoxName` **} {**

@HEIGHT { h **}**
@LEFT_WEB_THICKNESS { t_{lw} **}**
@RIGHT_WEB_THICKNESS { t_{rw} **}**
@TOP_LEFT_FLANGE_WIDTH { w_{tlf} **}**
@TOP_LEFT_FLANGE_THICKNESS { t_{tf} }
@TOP_LEFT_FLANGE_SKEW_ANGLE { α_{tlf} }
@TOP_FLANGE_REINFORCE_THICKNESS { t_{tr} }
@TOP_RIGHT_FLANGE_WIDTH { w_{trf} }
@TOP_RIGHT_FLANGE_SKEW_ANGLE { α_{trf} }
@BOTTOM_LEFT_FLANGE_WIDTH { w_{blf} }
@BOTTOM_LEFT_FLANGE_THICKNESS { t_{bf} }
@BOTTOM_LEFT_FLANGE_SKEW_ANGLE { α_{blf} }
@BOTTOM_FLANGE_REINFORCE_THICKNESS { t_{br} }
@BOTTOM_RIGHT_FLANGE_WIDTH { w_{brf} }
@BOTTOM_RIGHT_FLANGE_SKEW_ANGLE { α_{brf} }
@WEB_MATERIAL_NAME { `WebMaterialName` }
@TOP_REINFORCE_MATERIAL_NAME { `TopMaterialName` }
@BOTTOM_REINFORCE_MATERIAL_NAME { `BottomMaterialName` }
@IS_DEFINED_IN_FRAME { `FxdFrameName` }
@MESH_DENSITY { md }

}

}

2.7.5 Examples

A few examples that describe the construction procedure of this type of section are shown below.

Example 1
This example shows a rectangular box with skewed flanges. Here height, web thicknesses, flange widths, flange skew angles, materials and mesh density are assigned for constructing this section.This example also shows the principal centroidal axes of bending.

Example 2
This example shows a rectangular box with top and bottom flange reinforcements. Here height, web thicknesses, flange widths, flange thickness, bottom left flange skew angle, materials and mesh density are assigned for constructing this section. This example also shows the axial stress field over the cross-section under the applied sectional loads.

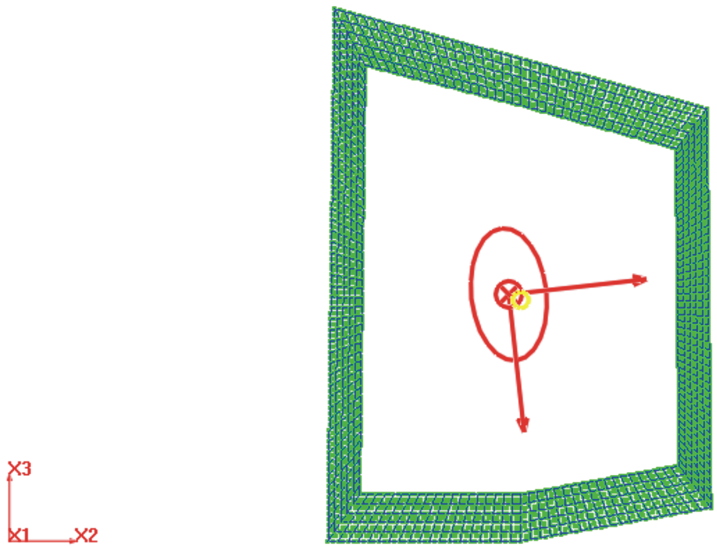

Figure 2.65: Example 1-Rectangular Box.

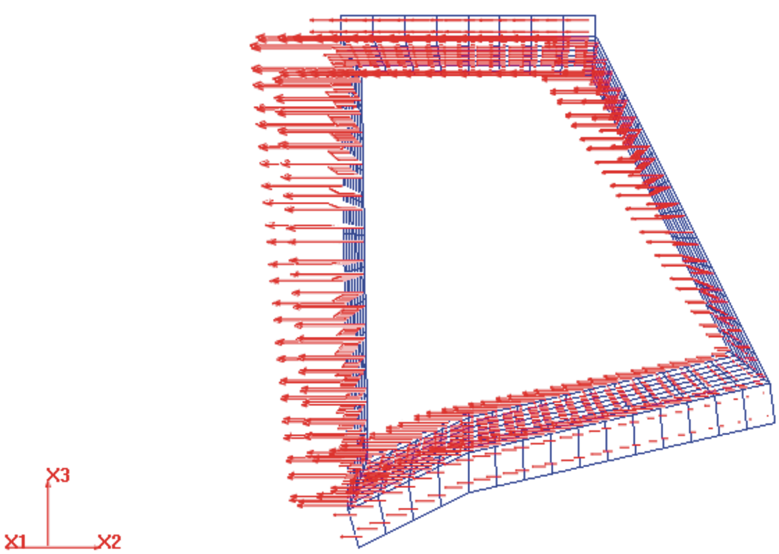

Figure 2.66: Example 2-Rectangular Box.

2.8 Definition of rectangular sections

Rectangular sections are parametric configurations of the shape depicted in fig. 2.67. They consist of a rectangular section, possibly reinforced by top and/or bottom flanges. The section consists of up to three three zones to which independent material properties can be assigned.

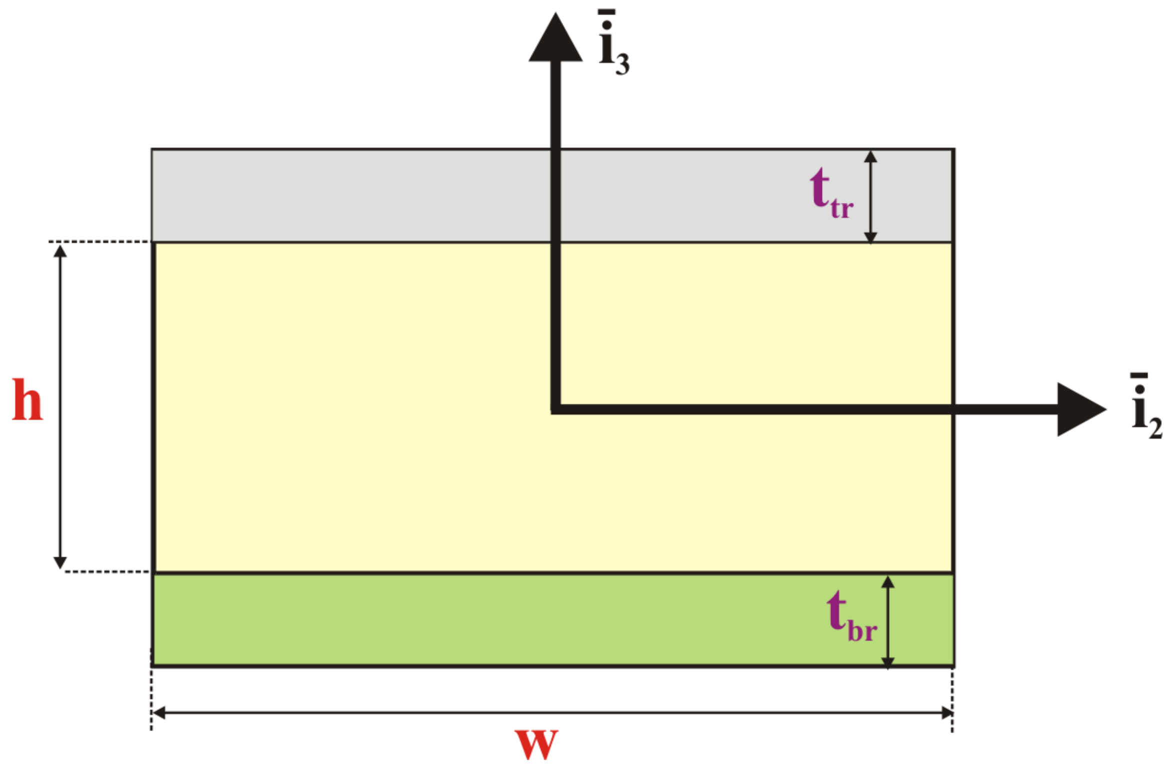

Figure 2.67: Configuration of the rectangular-Section.

Rectangular sections are defined by means of three dialog tabs.

1. The *Rectangular section* dialog tab as described in section 2.8.1, which defines the name of the section.
2. The *Dimensions* dialog tab as described in section 2.8.2, which defines the dimensions of the section.
3. The *Materials* dialog tab as described in section 2.8.3, which defines the materials the section is made of.

2.8.1 The *Rectangular section* dialog tab

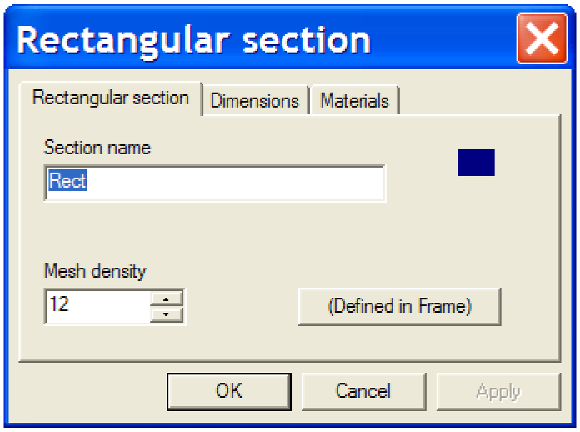

Figure 2.68: The *Rectangular section* dialog tab.

The *Rectangular section* dialog tab as described in section 2.68, defines the following data for the rectangular section.

1. **Section name.** Enter a unique name for the rectangular section.
2. **Mesh density.** Enter the desired mesh density as described in section 5.1, for the finite element discretization.
3. *(Defined in frame).* The geometry of the rectangular box can be defined with respect to a fixed frame as described in section 7.1, allowing translation and rotation of the section as a rigid body.

2.8.2 The *Dimensions* dialog tab

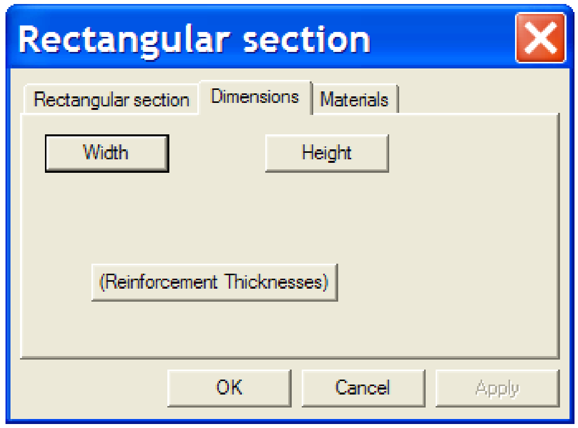

Figure 2.69: The *Dimensions* dialog tab.

The *Dimensions* dialog tab as described in fig. 2.62, defines the dimensions of the rectangular section shown in fig. 2.67. The dimensions of the section are defined by the following parameters.

1. The width, w, of the section (**required input**).
2. The height, h, of the section (**required input**).

   The thickness, t_{tr}, of the top reinforcement flange; this thickness applies to both left and right reinforcements that cannot exist independently of each other (*default value:* $t_{\mathrm{tr}} = 0$).
3. The thickness, t_{tr}, of the top reinforcement flange (*default value:* $t_{\mathrm{tr}} = 0$). This variable is also used as a flag for the presence of the top reinforcement flange: if $t_{\mathrm{tr}} \neq 0$, the top reinforcement flange is present.
4. The thickness, t_{br}, of the bottom reinforcement flange (*default value:* $t_{\mathrm{br}} = 0$). This variable is also used as a flag for the presence of the bottom reinforcement flange: if $t_{\mathrm{br}} \neq 0$, the bottom reinforcement flange is present.

2.8.3 The *Materials* dialog tab

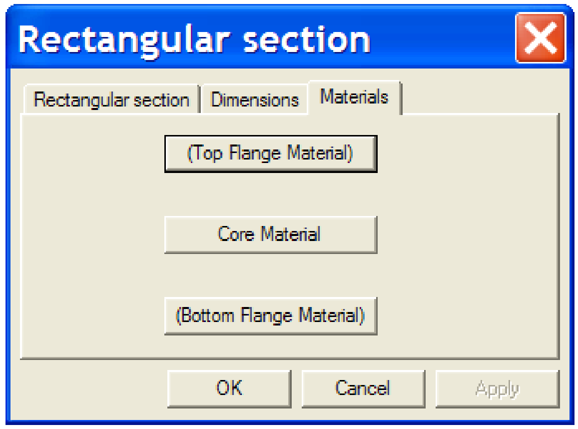

Figure 2.70: The *Materials* dialog tab.

The *Materials* dialog tab as described in fig. 2.70, defines the materials the rectangular section is made of. As shown in fig. 2.71, the section is divided into three zones.

1. The top reinforcement flange consists of a single component labeled *TFlgR*.
2. The central portion of the section consists of a single component labeled **Core**.
3. The bottom reinforcement flange consists of a single component labeled *BFlgR*.

It is possible to assign material properties as described in section 4.1, or solid properties as described in section 4.2, to each zone of the rectangular section.

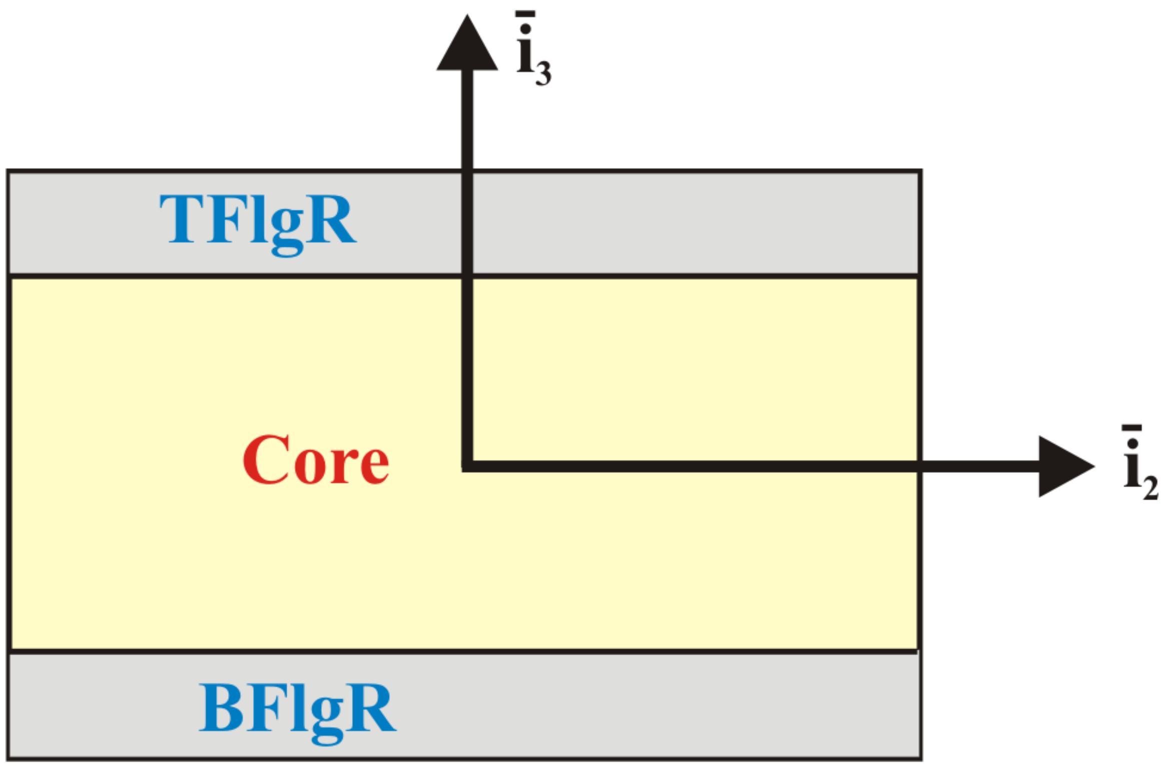

Figure 2.71: Solid zones of the rectangular section.

2.8.4 Formatted input

The data defined in the above sections will be saved in a specially formatted input file which has the following structure.

@RECTANGULAR_SECTION_DEFINITION {

@RECTANGULAR_SECTION_NAME { `TboxName` **} {**

@WIDTH { *w* **}**

@HEIGHT { *h* **}**

@CORE_MATERIAL_NAME { `CoreMaterialName` }

@TOP_REINFORCE_MATERIAL_NAME { `TopMaterialName` }

@BOTTOM_REINFORCE_MATERIAL_NAME { `BottomMaterialName` }

@IS_DEFINED_IN_FRAME { `FxdFrameName` }

@MESH_DENSITY { md }

}

}

2.8.5 Examples

A few examples that describe the construction procedure of this type of section are shown below.

Example 1

This example shows a rectangular section. Here width, height, frame, core material and mesh density are assigned for constructing this section. This example also shows the inverse of the reserve factor field over the cross- section under the applied sectional loads.

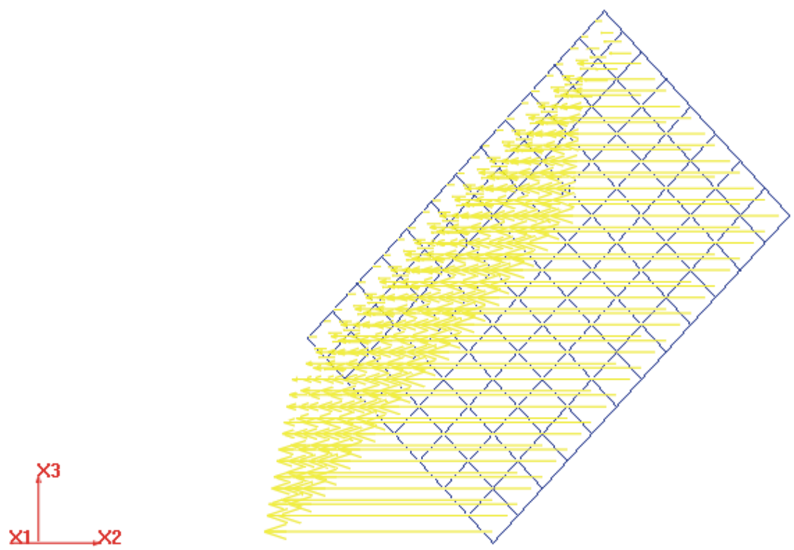

Figure 2.72: Example 1-Rectangular Section.

Example 2

This example shows also a rectangular section. Here width, height, frame, reinforcement thicknesses, composite core materials and mesh density are assigned for constructing this section. This example also shows the shear stress field over the cross-section under the applied sectional loads.

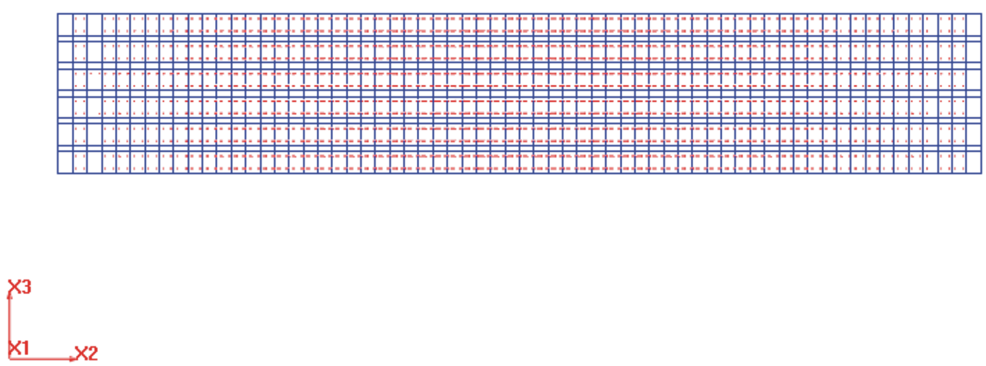

Figure 2.73: Example 2-Rectangular Section.

2.9 Definition of circular tubes

Circular tubes are predefined sections presenting the shape shown in fig. 2.74. Circular tubes consist of an area included between two circles. The section consists of a single zone to which material properties can be assigned. The circular tube is a **closed circular tube**, as shown in fig. 2.74. **Open circular tubes** can be defined with the help of the circular arc predefined section as described in section 2.2.

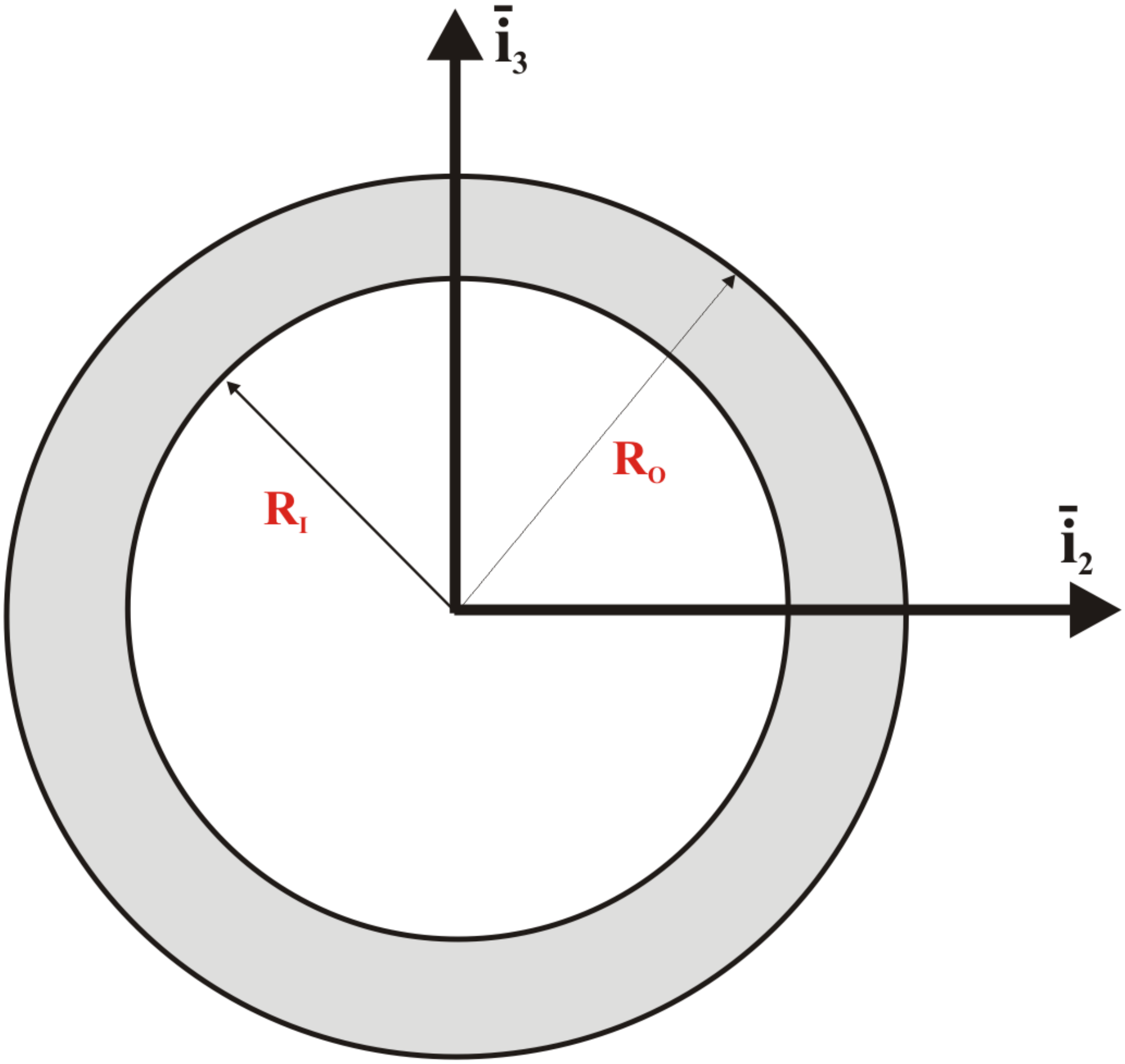

Figure 2.74: Configuration of the circular tube.

Circular tubes are defined by means of three dialog tabs.

1. The *Circular tube* dialog tab as described in section 2.9.1, which defines the name of the section.
2. The *Dimensions* dialog tab as described in section 2.9.2, which defines the dimensions of the section.
3. The *Materials* dialog tab as described in section 2.9.3, which defines the materials the section is made of.

2.9.1 The *Circular tube* dialog tab

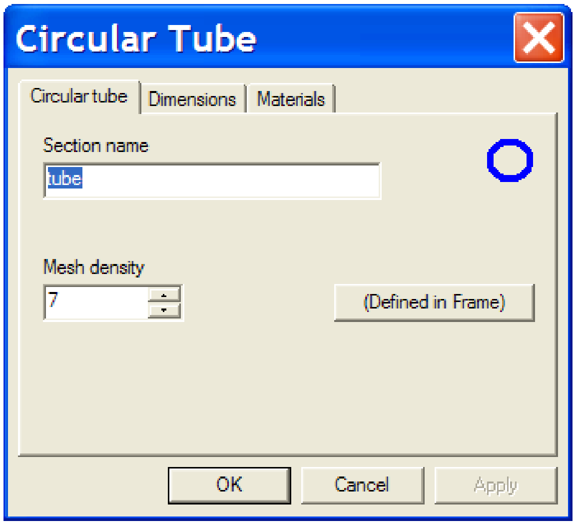

Figure 2.75: The *Circular tube* dialog tab.

The *Circular tube* dialog tab as described in fig. 2.75, defines the following data for the circular tube.

1. **Section name.** Enter a unique name for the *Circular Tube*.
2. **Mesh density.** Enter the desired mesh density as described in section 5.1, for the finite element discretization.
3. *(Defined in frame).* The geometry of the circular tube can be defined with respect to a fixed frame as described in section 7.1, allowing translation and rotation of the section as a rigid body.

2.9.2 The *Dimensions* dialog tab

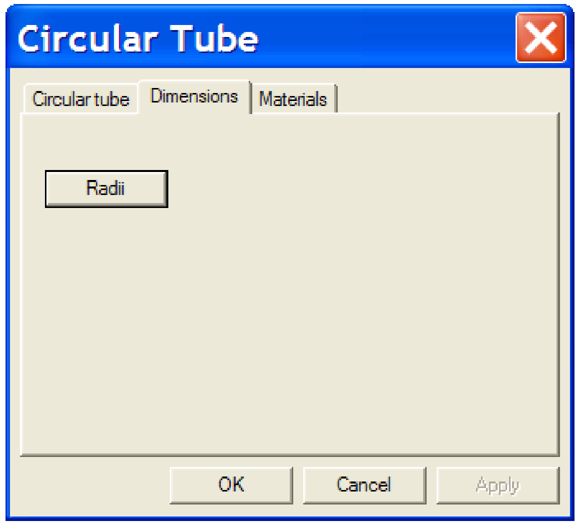

Figure 2.76: The *Dimensions* dialog tab.

The *Dimensions* dialog tab as described in fig. 2.76, defines the dimensions of the circular tube shown in fig. 2.74. The dimensions of the section are defined by two parameters.

1. The outer radius, R_O, of the circular tube (**required input**).
2. The inner radius, R_I, of the circular tube (**required input**).

2.9.3 The *Materials* dialog tab

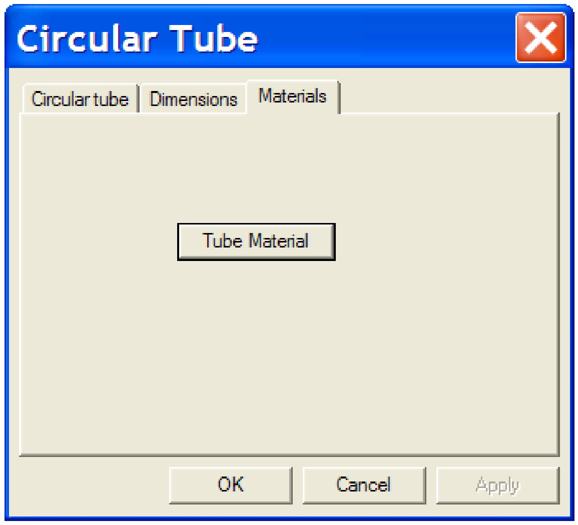

Figure 2.77: The *Materials* dialog tab.

The *Materials* dialog tab as described in fig. 2.77, defines the materials the circular tube is made of. It is possible to assign material properties as described in section 4.1, or solid properties as described in section 4.2, to each zone of the circular tube.

2.9.4 Formatted input

The data defined in the above sections will be saved in a specially formatted input file which has the following structure.

@CIRCULAR_TUBE_DEFINITION {

@CIRCULAR_TUBE_NAME { `TubeName` **} {**

@INNER_RADIUS { R_I **}**
@OUTER_RADIUS { R_O **}**
@MATERIAL_PROPERTY_NAME { `MaterialName` **}**
@IS_DEFINED_IN_FRAME { `FxdFrameName` }
@MESH_DENSITY { md }

}

}

2.9.5 Examples

A few examples that describe the construction procedure of this type of sections are show below.

Example 1

This example shows a circular tube section. Here inner and outer radii, frame, material and mesh density are assigned for constructing this section. This example also shows the warping displacement of the cross-section under the applied sectional loads.

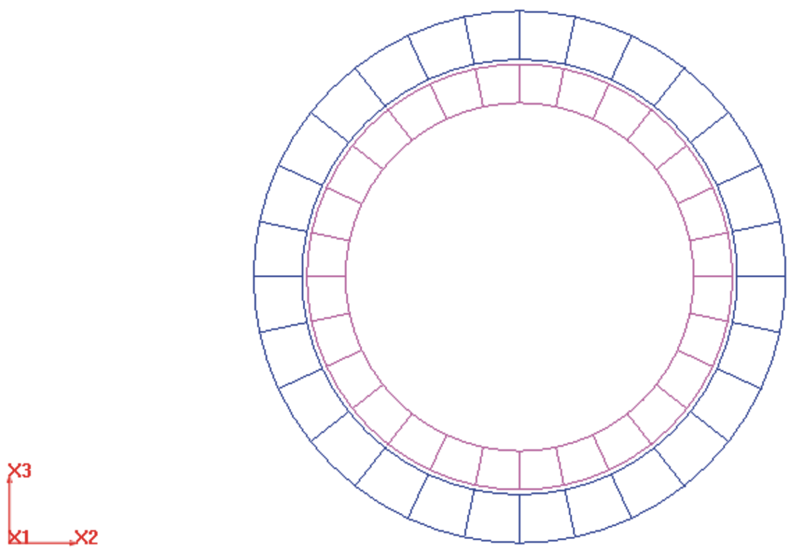

Figure 2.78: Example 1-Circular Tube.

Example 2

This example shows also a circular tube section. Here inner and outer radii, frame, materials property and mesh density are assigned for constructing this section. This example also shows the principal axes of inertia at the mass center.

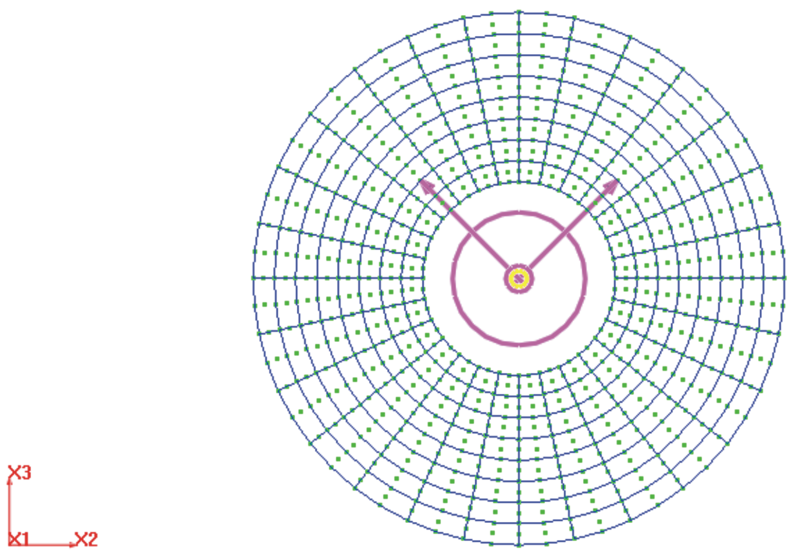

Figure 2.79: Example 2-Circular Tube.

2.10 Definition of triangular sections

Triangular sections are parametric configurations of the shape depicted in fig. 2.80. They consist of a triangular section, which can be open or closed. The section consists of a single zone which material properties can be assigned. Note that through the use of fixed frames as described in section 7.1, the triangular section can be made into the following shapes: △, ◁, ▷, or ▽.

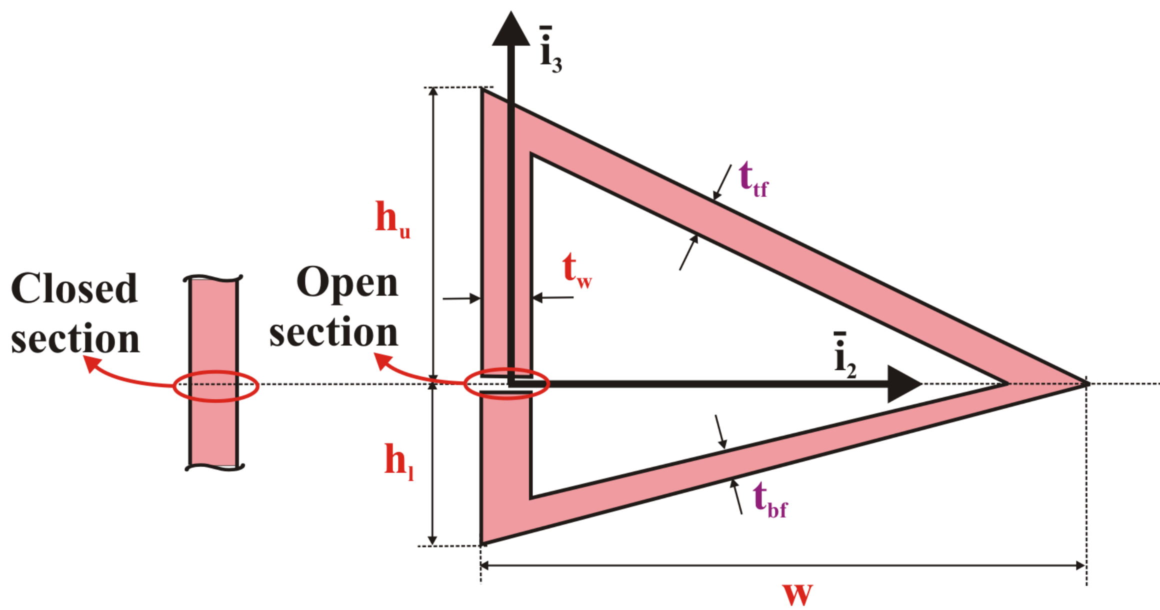

Figure 2.80: Configuration of the triangular section with open or closed section.

Triangular sections are defined by means of three dialog tabs.

1. The *Triangular section name* dialog tab as described in section 2.10.1, which defines the name of the section.
2. The *Dimensions* dialog tab as described in section 2.10.2, which defines the dimensions of the section.
3. The *Materials* dialog tab as described in section 2.10.3, which defines the materials the section is made of.

2.10.1 The *Triangular section name* dialog tab

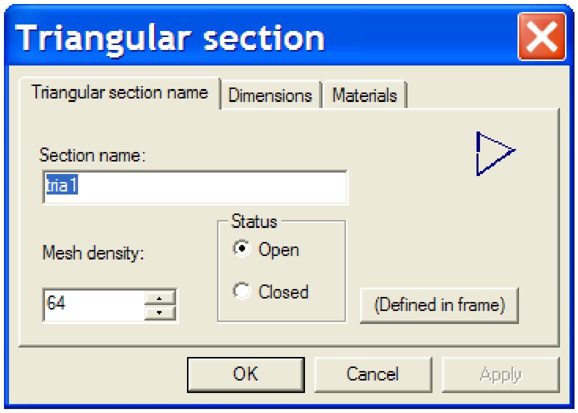

Figure 2.81: The *Triangular section name* dialog tab.

The *Triangular section name* dialog tab as described in fig. 2.81, defines the following data for the triangular section.

1. **Section name.** Enter a unique name for the triangular section.
2. **Section status.** Select an open or closed section.
3. **Mesh density.** Enter the desired mesh density as described in section 5.1, for the finite element discretization.
4. *(Defined in frame).* The geometry of the triangular section can be defined with respect to a fixed frame as described in section 7.1, allowing translation and rotation of the section as a rigid body.

2.10.2 The *Dimensions* dialog tab

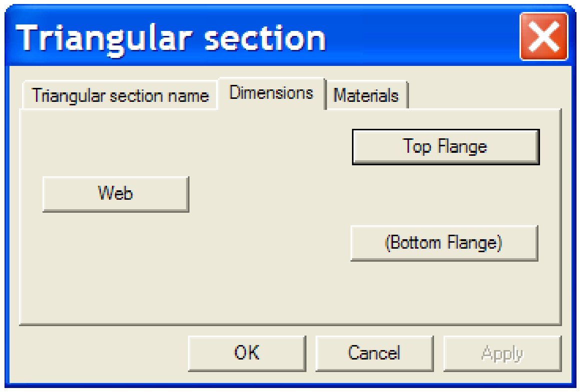

Figure 2.82: The Dimension dialog window.

The *Dimensions* dialog tab as described in fig. 2.82, defines the dimensions of the triangular section shown in fig. 2.80. The dimensions of the section are defined by the following parameters.

Web dimensions

1. The height, h_{u}, of the top web (**required input**).
2. The height, h_{b}, of the bottom web (**required input**).
3. The thickness, t_{w}, of the web (**required input**).

Top flange dimensions

1. The width, w, of the section (**required input**).
2. The thickness, t_{tf}, of the top flange (*default value:* $t_{\mathrm{tf}} = t_{\mathrm{w}}$).

Bottom flange dimensions

1. The width, w, of the section (**required input**).
2. The thickness, t_{bf}, of the bottom flange (*default value:* $t_{\mathrm{bf}} = t_{\mathrm{w}}$).

2.10.3 The *Materials* dialog tab

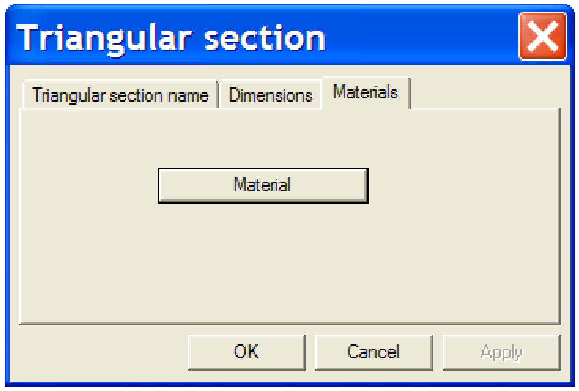

Figure 2.83: The *Materials* dialog tab.

The *Materials* dialog tab as described in fig. 2.83, defines the material the triangular section is made of. As shown in fig. 2.84, the section features a single zone.

1. The top reinforcement flange consists of the components labeled *TWeb*, *TFlg* *BFlg* and *BWeb*.

It is possible to assign material properties as described in section 4.1, or solid properties as described in section 4.2, to the triangular section.

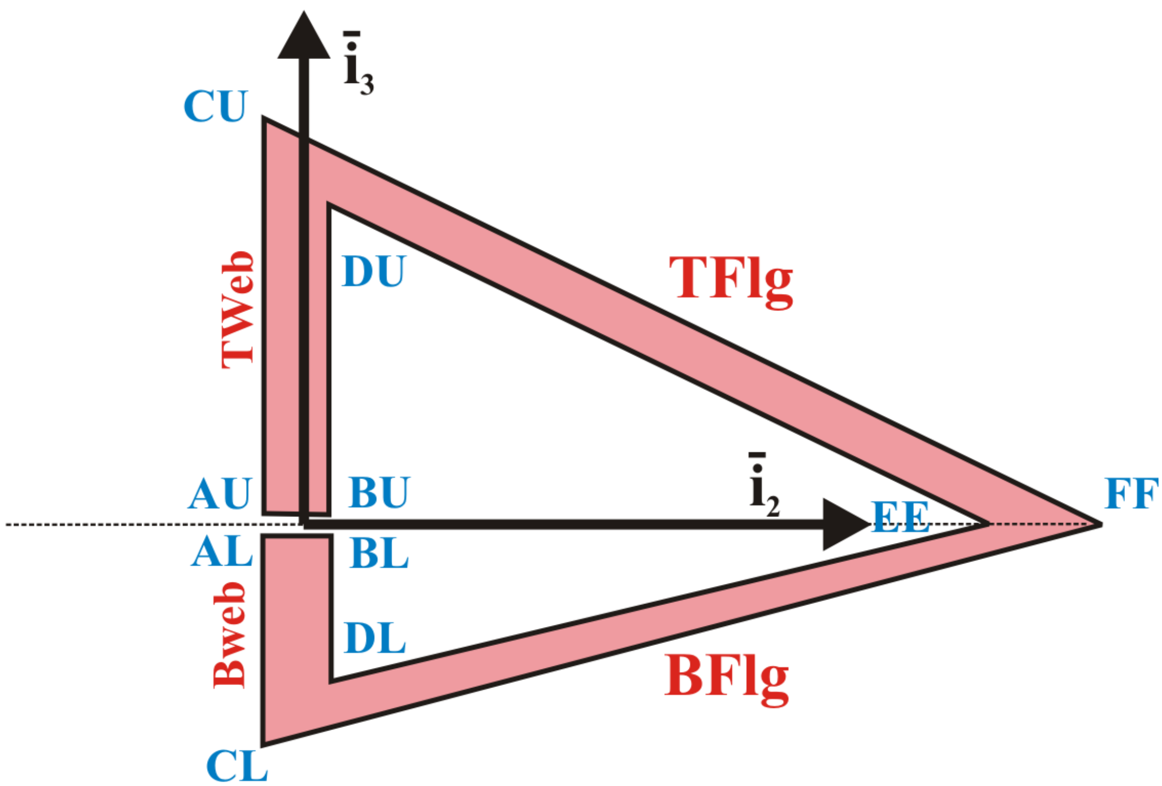

Figure 2.84: The three zones of the triangular section.

2.10.4 Formatted input

The data defined in the above sections will be saved in a specially formatted input file which has the following structure.

@TRIANGULAR_SECTION_DEFINITION {

@TRIANGULAR_SECTION_NAME { `TriaName` **} {**

@TOP_WEB_HEIGHT { h_{tw} **}**
@BOTTOM_WEB_HEIGHT { h_{bw} **}**
@WIDTH { `w` **}**
@WEB_THICKNESS { t_{w} }
@OPEN_SECTION { `YES/NO` }
@WEB_MATERIAL_NAME { $t_{\mathrm{WMaterialName}}$ }
@IS_DEFINED_IN_FRAME { `FxdFrameName` }
@MESH_DENSITY { md }

}

}

2.10.5 Examples

A few examples that describe the construction procedure of this type of section are shown below.

Example 1

This example shows an open triangular section. Here top and bottom web heights, width, web thickness, material, frame and mesh density are assigned for constructing this section. This example also shows the principal axes of shearing at the shear center.

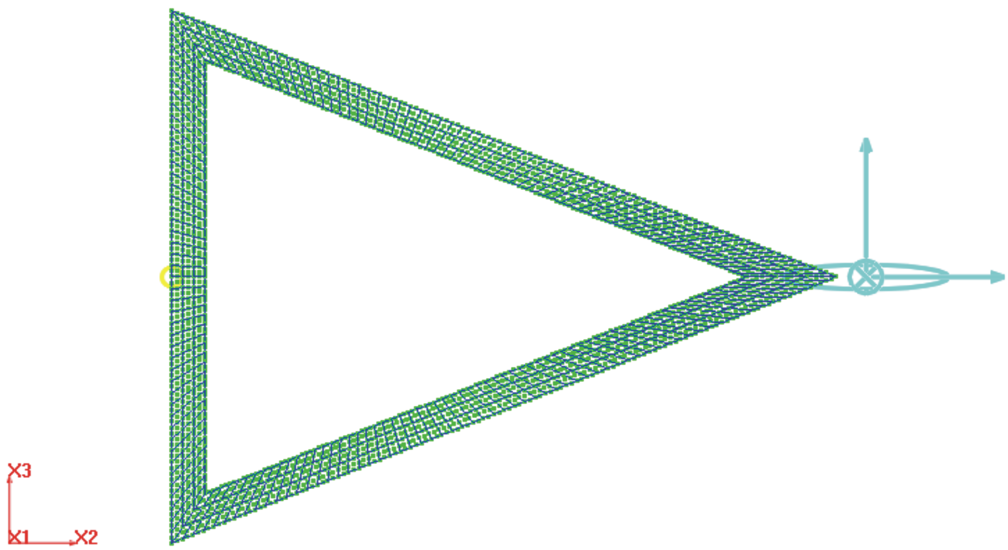

Figure 2.85: Example 1.Triangular Section

2.11 Definition of T-sections

T-sections are parametric configurations of the shape depicted in fig. 2.86. They consist of a T-section, possibly reinforced by a top flange. The section consists of up to three zones to which independent material properties can be assigned. Note that through the use of fixed frames as described in section 7.1, the T-section can be made into the following shapes: ⊥, ⊣, ⊢, or ⊤.

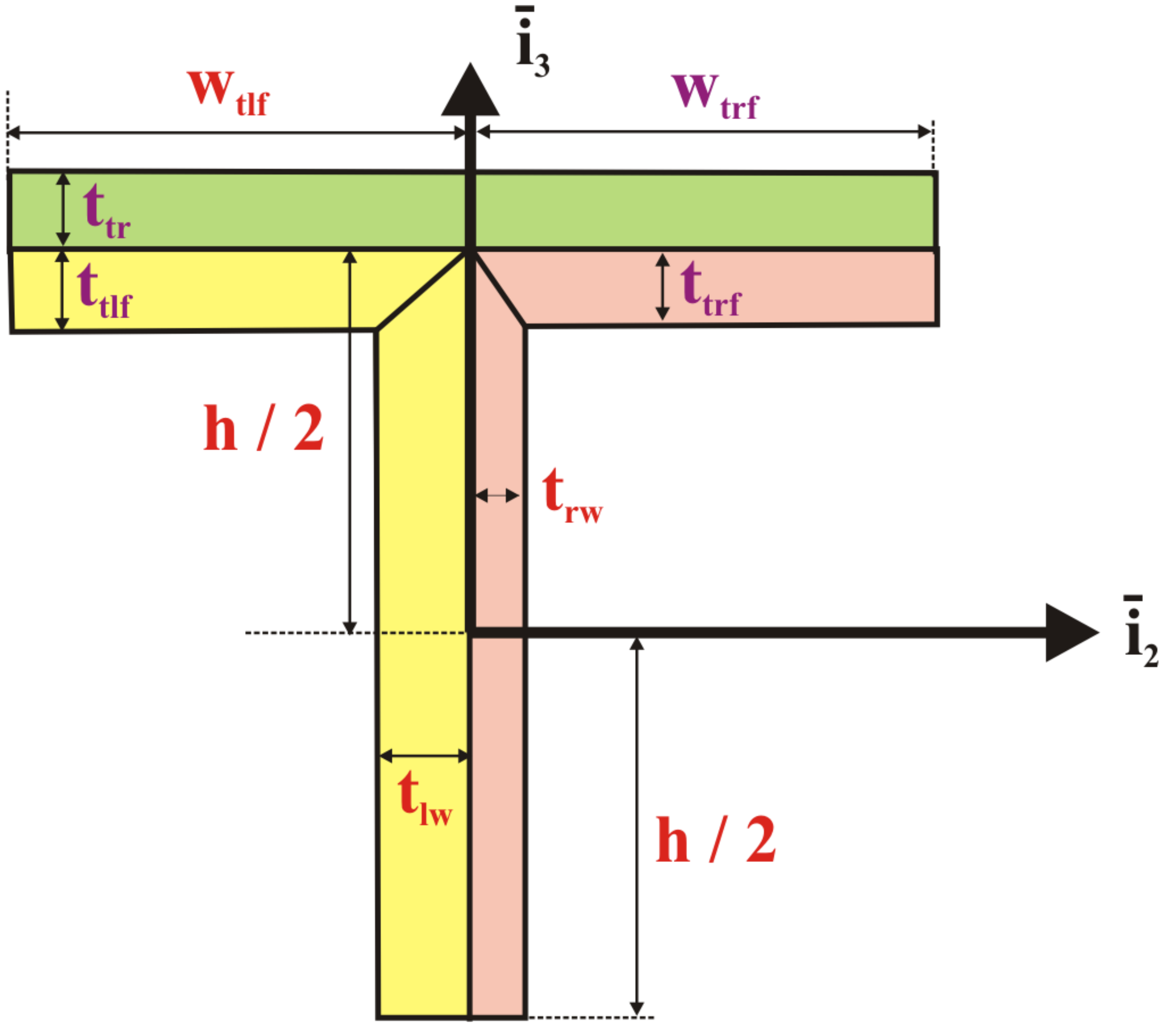

Figure 2.86: Configuration of the T-section.

T-sections are defined by means of three dialog tabs.

1. The *T-section* dialog tab as described in section 2.11.1, which defines the name of the section.
2. The *Dimensions* dialog tab as described in section 2.11.2, which defines the dimensions of the section.
3. The *Materials* dialog tab as described in section 2.11.3, which defines the materials the section is made of.

2.11.1 The *T-section* dialog tab

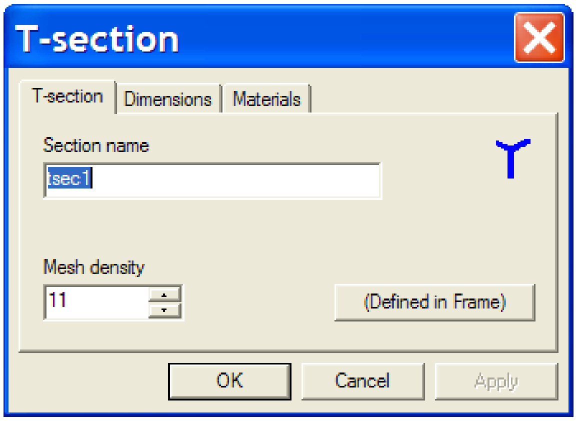

Figure 2.87: The *T-section* dialog tab.

The *T-section* dialog tab as described in fig. 2.87, defines the following data for the T-section.

1. **Section name.** Enter a unique name for the T-section.
2. **Mesh density.** Enter the desired mesh density as described in section 5.1, for the finite element discretization.
3. *(Defined in frame).* The geometry of the T-section can be defined with respect to a fixed frame as described in section 7.1, allowing translation and rotation of the section as a rigid body.

2.11.2 The *Dimensions* dialog tab

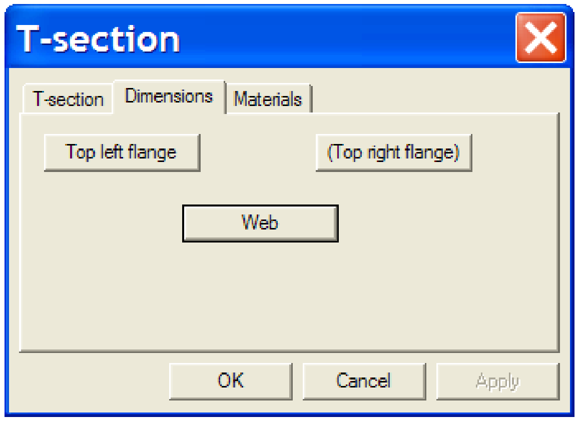

Figure 2.88: The Dimension dialog window.

The *Dimensions* dialog tab as described in fig. 2.88, defines the dimensions of the T-section shown in fig. 2.86. The dimensions of the section are defined by the following parameters.

Web dimensions

1. The height, h, of the section (**required input**).
2. The thickness, t_{lw}, of the left part of the web (**required input**).
3. The thickness, t_{rw}, of the right part of the web (**required input**).

Top left flange dimensions

1. The width, w_{tlf}, of the top left flange (**required input**).
2. The thickness, t_{tlf}, of the top left flange (*default value:* $t_{\mathrm{tlf}} = t_{\mathrm{lw}}$).
3. The skew angle, α_{tlf}, of the top left flange, positive up, measured in degrees (*default value:* $\alpha_{\mathrm{tlf}} = 0$).
4. The thickness, t_{tr}, of the top reinforcement flange; this thickness applied to both left and right reinforcements that cannot exist independently of each other (*default value:* $t_{\mathrm{tr}} = 0$). This variable is also used as a flag for the presence of the top flange reinforcement: if $t_{\mathrm{tr}} > 0$, this reinforcement is present.

Top right flange dimensions

1. The width, w_{trf}, of the top right flange (*default value:* $w_{\mathrm{trf}} = w_{\mathrm{tlf}}$).
2. The thickness, t_{trf}, of the top right flange (*default value:* $t_{\mathrm{trf}} = t_{\mathrm{rw}}$).
3. The skew angle, α_{trf}, of the top right flange, positive up, measured in degrees (*default value:* $\alpha_{\mathrm{trf}} = 0$).

2.11.3 The *Materials* dialog tab

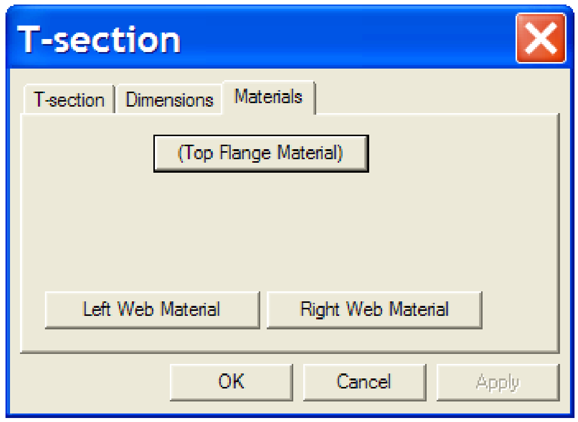

Figure 2.89: The *Materials* dialog tab.

The *Materials* dialog tab as described in fig. 2.89, defines the materials the T-section is made of. As shown in fig. 2.90, the section is divided into three zones.

1. The top reinforcement flange consists of the components labeled *TLFlgR* and *TRFlgR*.
2. The left portion of the T-section consists of the components labeled **TLFlg** and **LWeb**.
3. he left portion of the T-section consists of the components labeled **TRFlg** and **RWeb**.

It is possible to assign material properties as described in section 4.1, or solid properties as described in section 4.2, to each zone of the T-section.

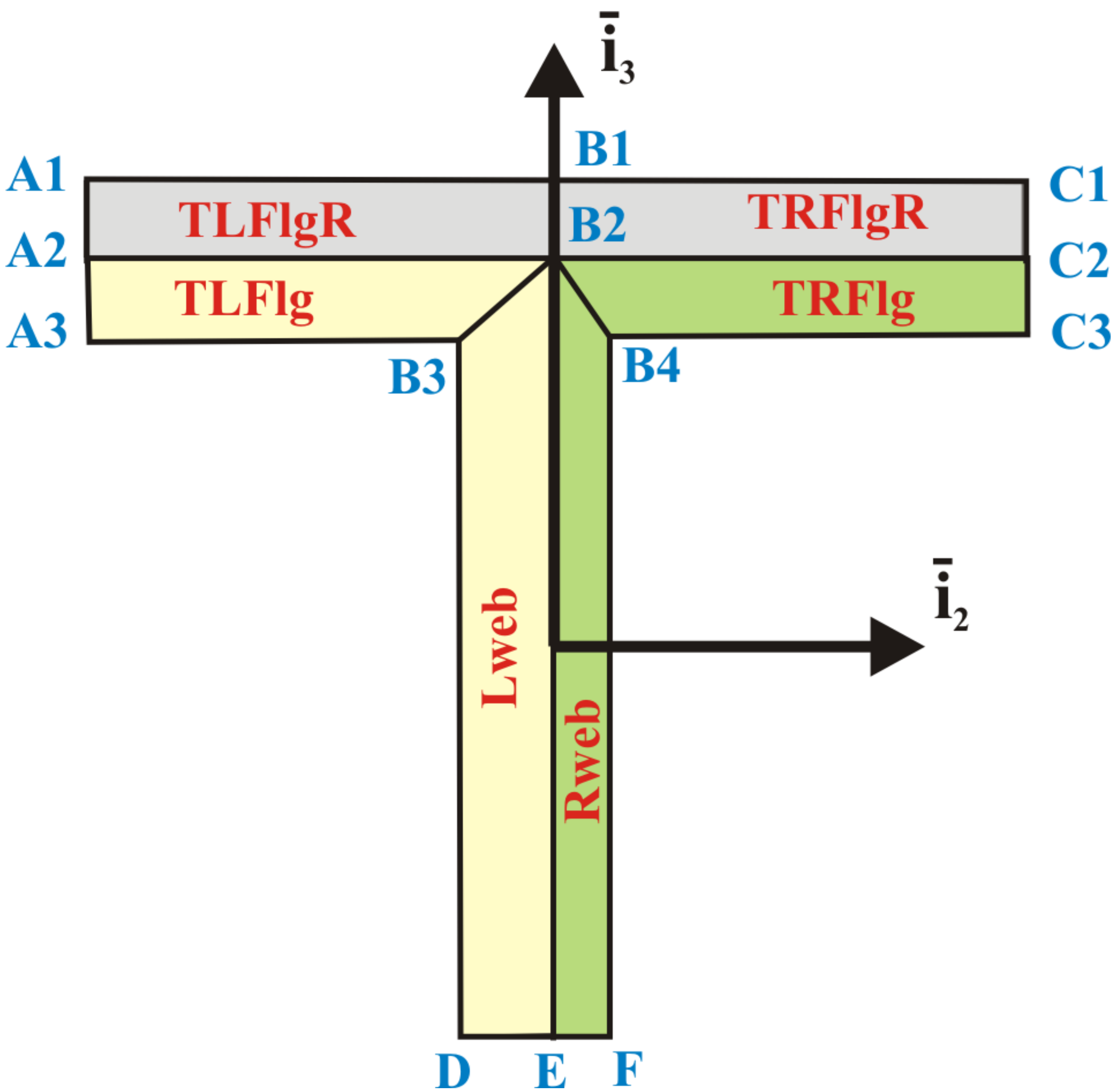

Figure 2.90: The three zones of the T-section.

2.11.4 Formatted input

The data defined in the above sections will be saved in a specially formatted input file which has the following structure.

@T_SECTION_DEFINITION {

@T_SECTION_NAME { `TsecName` **} {**

@WEB_HEIGHT { h **}**
@LEFT_WEB_THICKNESS { t_{lw} **}**
@RIGHT_WEB_THICKNESS { t_{rw} **}**
@TOP_LEFT_FLANGE_WIDTH { w_{tlf} }
@TOP_LEFT_FLANGE_THICKNESS { t_{tlf} }
@TOP_FLANGE_REINFORCE_THICKNESS { t_{tr} }
@LEFT_WEB_MATERIAL_NAME { `LWebMaterialName` **}**
@RIGHT_WEB_MATERIAL_NAME { `RWebMaterialName` **}**
@TOP_REINFORCE_MATERIAL_NAME { `RMaterialName` **}**
@IS_DEFINED_IN_FRAME { `FxdFrameName` }
@MESH_DENSITY { md }

}

}

2.11.5 Examples

A few examples that describe the construction procedure of this type of section are shown below.

Example 1

This example shows a T-section with skewed flange and top reinforcement. Here web height, web thicknesses, flange widths, flange thicknesses,flange skew angles, materials and mesh density are assigned for constructing this section. This example also shows the inverse of the reserve factor field over the cross-section under the applied sectional loads.

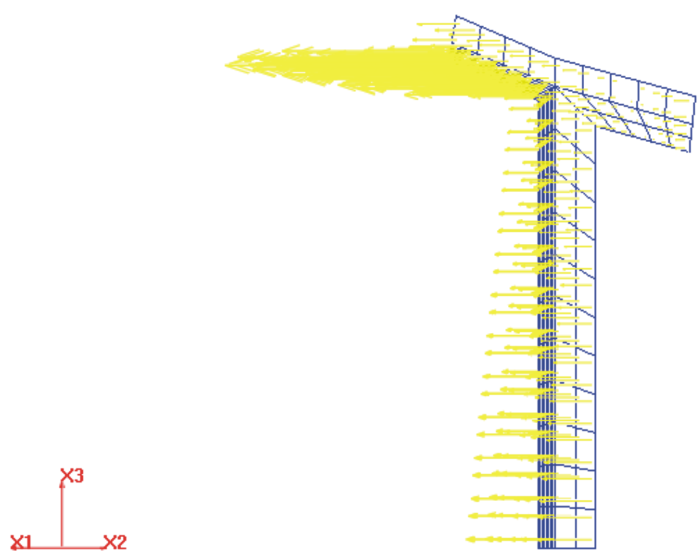

Figure 2.91: Example 1.T-section

Example 2

This example shows a T-section with flange reinforcement. Here web height, web thicknesses, flange widths, flange thicknesses, materials and mesh density are assigned for constructing this section. This example also shows the axial stress field over the cross-section under the applied sectional loads.

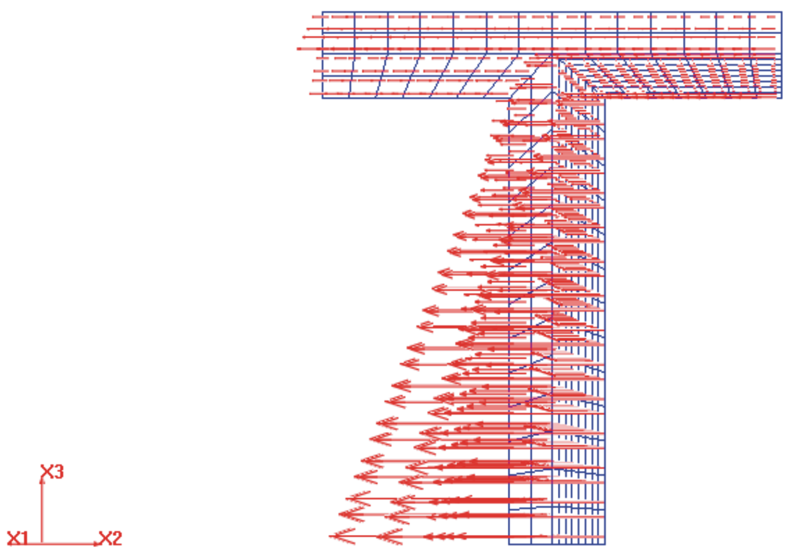

Figure 2.92: Example 2.T-section

Chapter 3

Builder

3.1 Introduction

Composite materials have found increasing use in aerospace and civil engineering constructions. Typically, complex sections are build by laying layers of composite materials in a mold; for lower performance structures, extrusion can be used. A typical beam section will be viewed here as a number of interconnected walls, as depicted in fig. 3.1. The upper flange of the profile consists of `Wall1` and `Wall2`, whereas the lower flange consists of `Wall4` and `Wall5`. The web consists of `Wall3`, and finally, the trailing edge tab consists of `Wall6`. The web and flanges are connected together by T-connectors, labeled `Tcon1` and `Tcon2`. The trailing edge tab splits into the upper and lower flanges through a split connector, labeled `Split1`. Finally, two walls can be directly connected to each other, such as `Wall2` and `Wall4` near the leading edge of the airfoil.

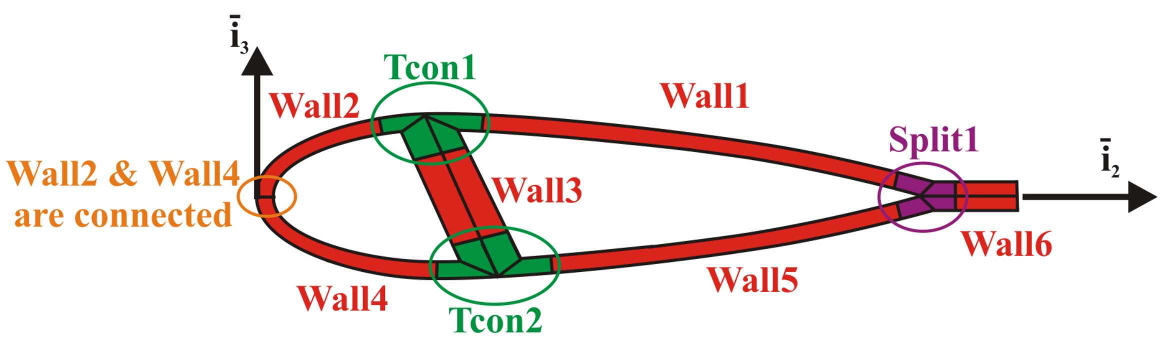

Figure 3.1: Airfoil Section construction

The definition of sections will involve the following components

1. Walls, see section 3.2.
2. Split-connectors, see section 3.3.
3. T-connectors, see section 3.4.
4. V-connectors, see section 3.5.

3.1.1 Examples

An example that describes the construction procedure of this type of section is shown below.

Example 1

This example constructs a simple airfoil section of NACA 4 digit series, which has one web. This section has five walls, two Tcon and one Vcon connectors. Wall1 and Wall5 have two layers. Wall2, Wall3 and Wall4

have more than two layers with ply-add/drop. Wall2 and Wall4 are connected between them. This example also shows the principal centroidal axes of bending.

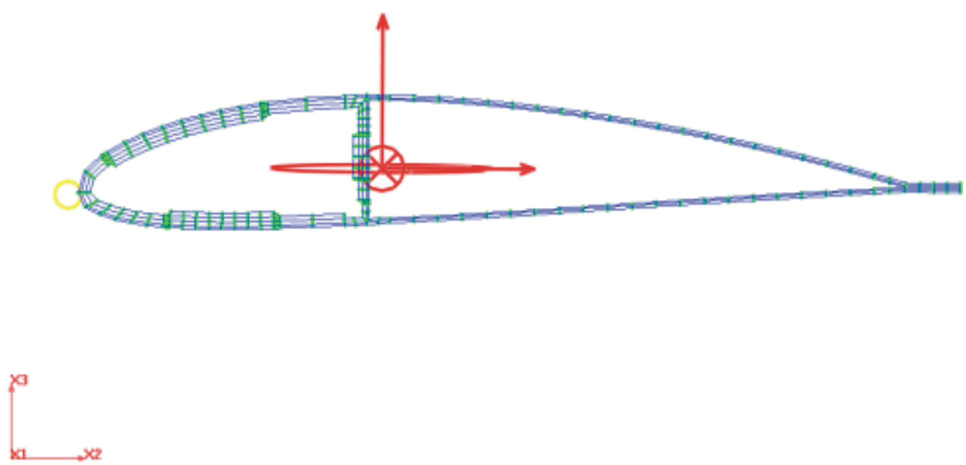

Figure 3.2: Example 1-Airfoil Section

3.2 Definition of walls

As discussed in section 1, a general beam section is defined as an assembly of interconnected walls, see fig. 3.1.

3.2.1 Wall geometry

The geometry of a wall is is defined by an oriented planar curve, as shown in fig. 3.3, defined in terms of its NURBS representation [2, 3]. Each point on the curve is associated with a variable η; the curve is defined between points **A** and **B**, corresponding to $\eta = 0$ and $\eta = 1$, respectively. Along the curve, the unit tangent and normal vectors, denoted $\bar{t}$ and $\bar{n}$, respectively, are defined such that $\bar{t}$ points in the direction of increasing values of η, and unit vectors $(\bar{t}, \bar{n})$ correspond to a planar rotation of unit vector $(\bar{\imath}_2, \bar{\imath}_3)$. It now becomes possible to define the upper and lower portion of the wall: the upper portion of the wall is located *above the curve*, *i.e.* in the direction of $\bar{n}$, whereas the lower portion of the wall is located *below the curve*.

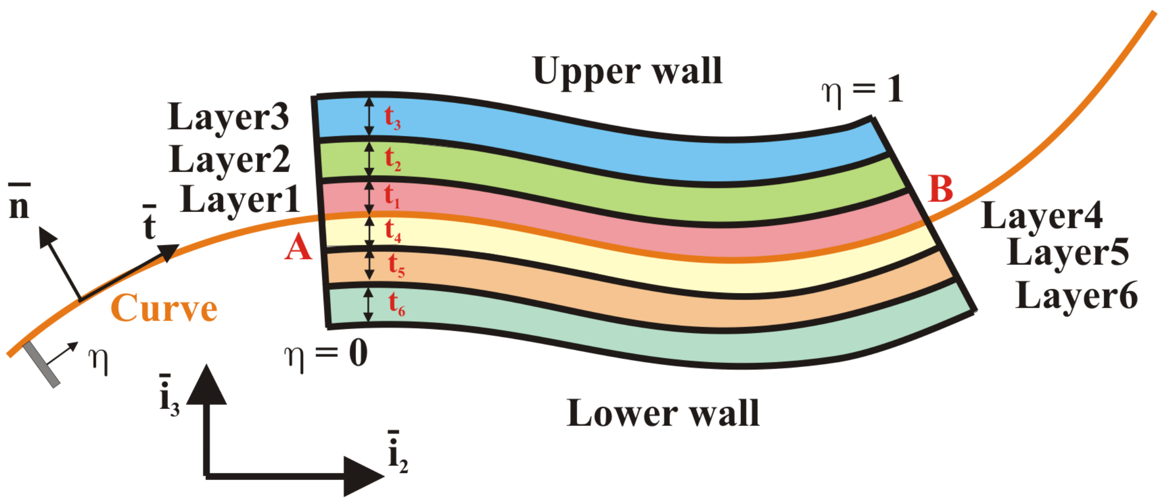

Figure 3.3: General configuration of a wall.

3.2.2 Wall stacking sequence

The upper and lower walls consist of a number of layers stacked on top and below the curve, respectively. Fig. 3.3 shows an upper wall consisting of three layers denoted `Layer1`, `Layer2` and `Layer3`, whereas the lower wall consists of layers `Layer4`, `Layer5` and `Layer6`. Each layer has a specific thickness; for instance, layers `Layer2` and `Layer3` are of thickness t_2 and t_3, respectively.

Layers do not necessarily span the entire length of the curve defining the geometry of the wall. In general, each layer starts at a given location, η_i, along the curve and ends at a location, η_f. The left portion of fig. 3.4 shows a typical configuration for a stacking sequence: layers `Layer1`, `Layer2` and `Layer5` have the same beginning and end coordinates, $\eta_i = 0$ and $\eta_f = 1$, respectively, whereas `Layer3` features $\eta_i = \eta_2$ and $\eta_f = \eta_3$ and `Layer4` features $\eta_i = \eta_1$ and $\eta_f = \eta_4$. Of course, when layers are dropped or added, the layers at the further distance from the curve defining the geometry of the wall "drop" onto the remaining layers, as illustrated on the right portion of fig. 3.4. An "add-drop" zone length defines the distance over which the layers are allowed to collapse onto a lower position.

Each layer is characterized by the following parameters *(1)* the layer name; *(2)* the layer beginning and end coordinates, η_i and η_f, respectively; *(3)* the layer thickness; *(4)* the material the layer is made of.

For transversely-isotropic materials, which are common for fiber reinforced composites, are easier to express materials properties as solid properties.

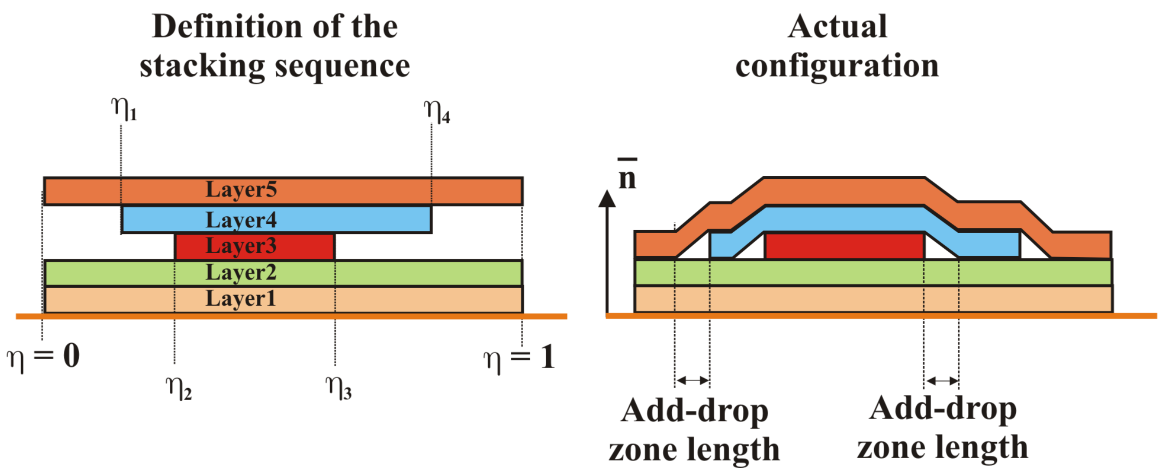

Figure 3.4: Definition of the stacking sequence and actual configuration of the wall.

Walls are allowed for ply-add/drop at any position between U=0 to U=1, as depicted in fig. 3.4. This ply-add/drop feature gives the freedom for designing a realistic airfoil section after connecting several walls with various connectors (i.e., Tcon, Vcon, Split etc).

Two walls with arbitrary lay-ups can be connected using "@CONNECTED AT 0" and "@CONNECTED AT 1" command. For connecting the initial position of a wall with the final position of another wall, "@CONNECTED AT 0" command is used. On the other hand, "@CONNECTED AT 1" command is considered for connecting the final position of a wall with the initial position of another wall, as depicted in fig. 3.5. The number of layers for walls must be equal at the connection. The layer thickness and material properties should also be compatible at the connection position of two walls.

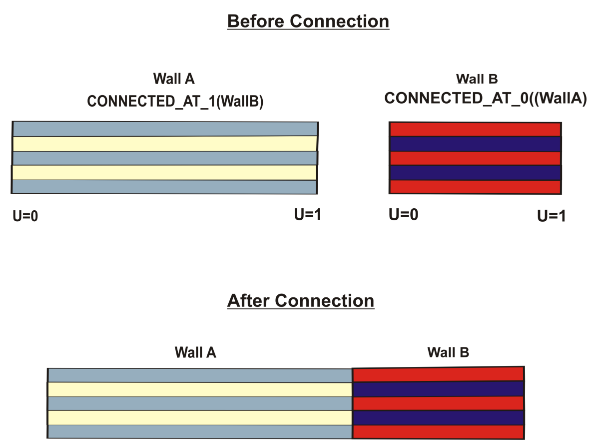

Figure 3.5: Connection of two Walls

3.2.3 Formatted input

The data defined in the above sections will be saved in a specially formatted input file which has the following structure.

@WALL_SECTION_DEFINITION {

@WALL_SECTION_NAME { `WallName` **} {**

@UPPER/LOWER_WALL_DEFINITION {

@LAYER_DEFINITION {
@LAYER_NAME { `LayerName1` **}**
@INITIAL_STATION { eta0 **}**
@FINAL_STATION { eta1 **}**
@LAYER_THICKNESS { t_{Layer1} **}**
@LAYER_MATERIAL{ `MaterialName` **}**
@LAYER_NAME { `LayerName2` **}**
@INITIAL_STATION { eta0 **}**
@FINAL_STATION { eta1 **}**
@LAYER_THICKNESS { t_{Layer2} **}**
@LAYER_MATERIAL{ `MaterialName` **}**
@LAYER_NAME { `LayerName3` **}**
@INITIAL_STATION { eta0 **}**
@FINAL_STATION { eta1 **}**
@LAYER_THICKNESS { t_{Layer3} **}**
@LAYER_MATERIAL{ `MaterialName` **}**
`Similarly add all other layers information here`
}
@CONNECTED_AT _1{ `WallName` **}**
@CONNECTED_AT _0{ `WallName` **}**
}
@CURVE_NAME{ `CurveName` **}**
@ADD _DROP_ZONE_SIZE{ delta **}**
@MESH_DENSITY{ n_{md} }

}

}

3.2.4 Examples

A few examples that describe the construction procedure of this type of section are shown below.

Example 1

This example shows a wall that has five layers. Each layer may have different materials. Ply-add/drop is also shown here. The mesh density can also be adjusted base on the requirement. This example also shows the axial stress field over the cross-section under the applied sectional loads.

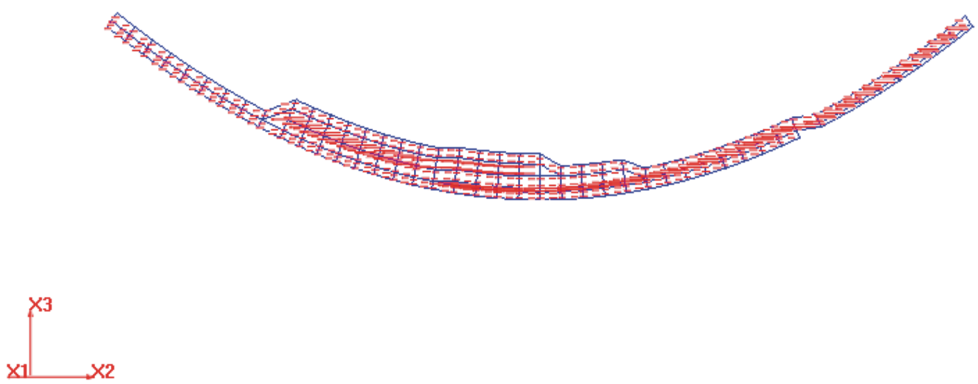

Figure 3.6: Example 1-Wall with arbitrary lay-ups.

Example 2

This example mainly shows the connection between two walls. Both walls have equal number of layers at the connection. The material properties of the layers need to be compatible at the joining position. The tangents of the base curves of the walls should also be equal or comparable (the angle between them should be less than 10 degrees). Both walls may have ply-add/drop. Ply-add/drop is not allowed at the connection position. This example also shows the principal axes of shearing at the shear center.

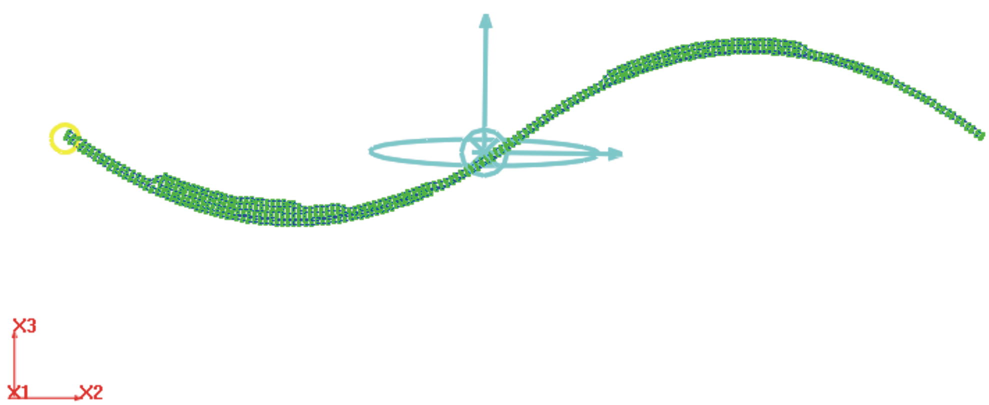

Figure 3.7: Example 2-Connection between two Walls.

3.3 Definition of Split sections

Split is a connector for connecting three walls. Three walls (Wall A, Wall B and Wall C) connected with Spilt, are depicted in fig. 3.8. Split can be constructed at any direction. i.e., one wall can split in two walls or two walls can marge into one wall. User has to provide the Wall A(connected at position A of Split), Wall B(connected at position B of Split) and Wall C(connected at position C of Split) information for constructing a Split.

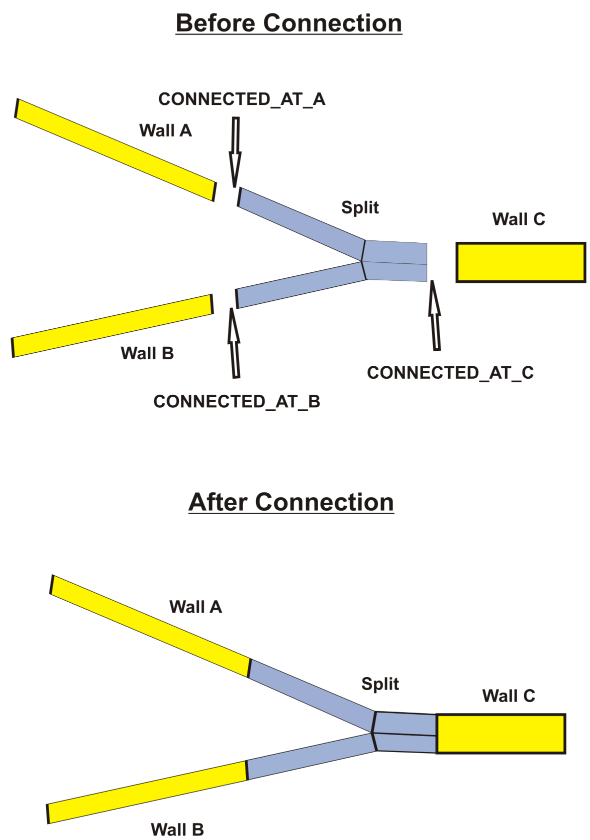

Figure 3.8: Three walls can be connected as above with a Split

The detail description of Split is shown in fig. 3.9. Basically wall information is used for Split construction. Points Ki, K1i, Mi and M1i (i= 1,2,3 ...n) are generated from WallA, WallB and WallC. Points Ki and Mi (or M1i) are the starting positions (at U=0) of various layers of WallA and WallC respectively. Similarly, points K1i are the final position (at U=1) of the layers of WallB. Li are the intersection points of the tangents of the layers of WallA (at U=0) and WallC (at U=0). L1i are the intersection points of the tangents of the layers of WallB (at U=1) and WallC (at U=0). Here the sum of the layers for WallA and WallB is equal to that of WallC. Curves and surfaces are constructed from Ki, K1i, Li, L1i, Mi and M1i. Solids are generated using the material and mesh properties. From WallB and lower part of WallC properties, LeftTailLo and RightTailLo sections are generated. LeftTailUp and RightTailUp sections are generated from WallA and upper part of WallC properties.

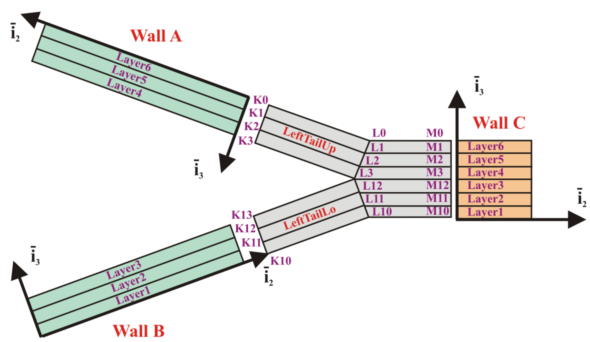

Figure 3.9: Three walls and Split(in detail)

3.3.1 Formatted input

The data defined in the above sections will be saved in a specially formatted input file which has the following structure.

@SPLIT_CONNECTOR_DEFINITION {

@SPLIT_CONNECTOR_NAME { `SplitName` **} {**

@CONNECTED_AT _A{ `WallAName`**}**
@CONNECTED_AT _B{ `WallBName`**}**
@CONNECTED_AT _C{ `WallCName`**}**

}

}

3.3.2 Examples

An example that describes the construction procedure of this type of section is shown below.

Example 1

This example shows a Split connector that connects 3 walls. WallA, WallB and WallC have 2, 2 and 4 layers respectively. Here the sum of the number of layers for WallA and WallB are equal to that of WallC at the connection position. This example also shows the principal centroidal axes of bending.

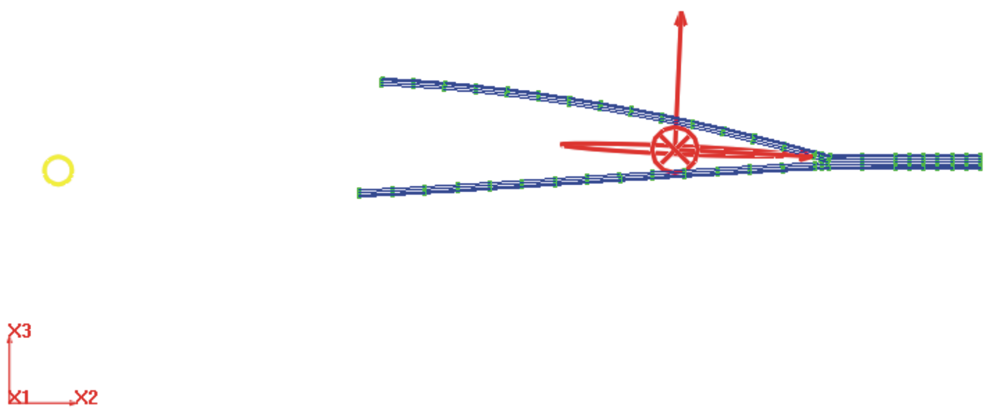

Figure 3.10: Example 1-Split connector with three Walls.

3.4 Definition of Tcon sections

Tcon is a connector for connecting three walls like a T-shape. Three walls (Wall A, Wall B and Wall C) connected with Tcon, are depicted in fig. 3.11, fig. 3.12. Tcon has three parts and those are web, right flange and left flange. The web is constructed from the layers of the Wall C. Wall C has layers at upper and lower directions. Upper direction means layers are stacked at the positive normal direction of the base curve of the wall. Whereas, lower direction means the layers are stacked at the negative normal direction. Right portion of the web (RWeb) and lower part of the right flange (TRFlg) are constructed from the upper direction layers of Wall C. Where as, the left part of the web (LWeb) and lower portion of the left flange (TLFlg) is constructed from the lower direction layers of the Wall C. The upper portion of the left and right flanges (TLFlgR and TRFlgR) are constructed from the pass through layers. Pass through layers are the common number of layers from the Wall A and Wall B. User has to provide the Wall A(connected at position A of Tcon), Wall B(connected at position B of Tcon), Wall C(connected at position C of Tcon) and Number of pass through layers information for constructing a Tcon.

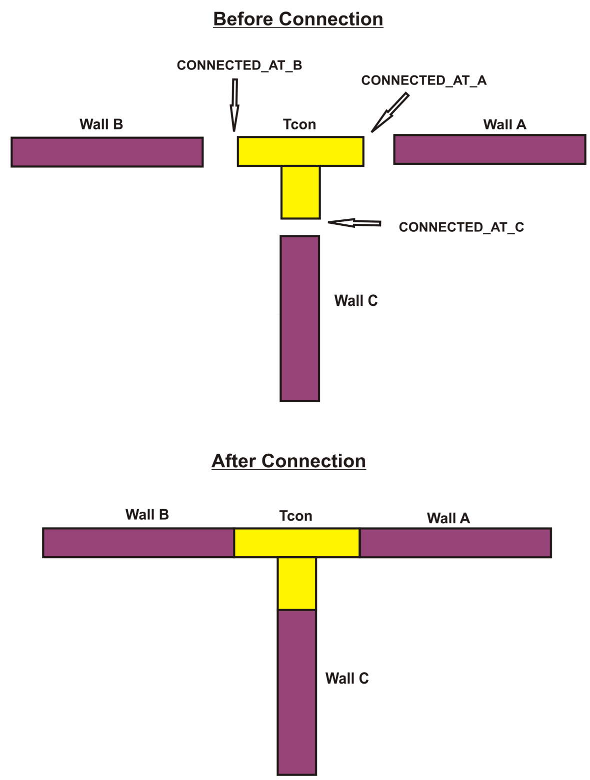

Figure 3.11: Three walls can be connected as above with a Tcon

The detail description of Tcon is shown in fig. 3.13 and fig. 3.14.Wall information is used for Tcon construction. Tcon can be orientated at upward and downward directions. For upward Tcon (shown in fig. 3.13), points Ai and Bi (i= 1,2,3 ....n) are generated from the pass through layers information of WallA (at U=0) and WallB (at U=1). Di are the middle points of Ai and Bi. Aai and Bbi (i= 1,2,3 ....n) are generated from the rest of the layers information of WallA (at U=0) and WallB (at U=1). Ci and Cci are constructed from the upper and lower parts of WallC (at U=0) respectively. Ei are the intersection points of the tangents of the layers of WallA (at U=0) and upper part of WallC (at U=0). Fi are the intersection points of the tangents of the layers of WallB (at U=1) and lower part of WallC (at U=0). Curves and surfaces are constructed using these points. Finally solid is defined using the material and mesh properties. TRFlgR and TLFlgR are constructed form the pass through layers information of WallA and WallB respectively. TRFlg and TLFlg are constructed form the other layers information of WallA and WallB. Where as, RWeb and LWeb are constructed form the upper and lower parts of WallC information respectively.

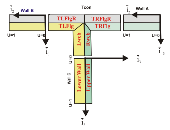

Figure 3.12: Three walls can be connected as above with a Tcon

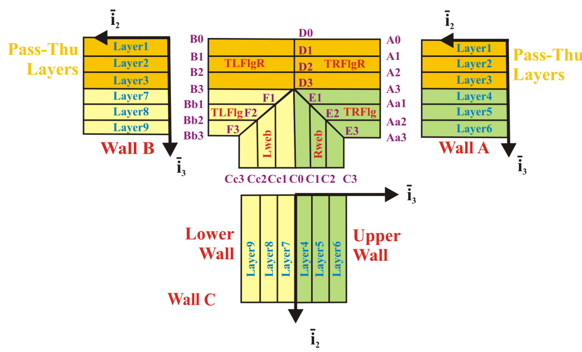

Figure 3.13: Three walls and Tcon in upward direction(in detail)

For downward Tcon (shown in fig. 3.14), points Ai and Bi (i= 1,2,3 ....n) are generated from the pass through layers of WallA (at U=1) and WallB (at U=0) . Like before, Di are the middle points of Ai and Bi. Aai and Bbi (i= 1,2,3 ....n) are generated from the rest of the layers information of WallA (at U=1) and WallB (at U=0). Ci and Cci are constructed from lower and upper parts of WallC (at U=1) respectively. Ei are the intersection points of the tangents of the layers of WallA (at U=1) and lower part of WallC (at U=1). Fi are the intersection points of the tangents of the layers of WallB (at U=0) and upper part of WallC (at U=0). Curves and surfaces are constructed using these points. Finally solid is defined using the material and mesh properties. TRFlgR and TLFlgR are constructed form the pass through layers information of WallA and WallB respectively. TRFlg and TLFlg are constructed form the other layers information of WallA and WallB. Where as, RWeb and LWeb are constructed form the lower and upper parts of WallC information respectively.

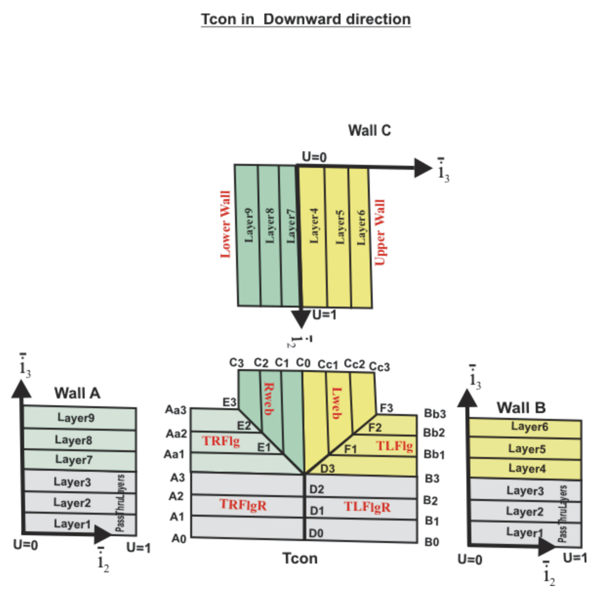

Figure 3.14: Three walls and Tcon in downward direction(in detail)

3.4.1 Formatted input

The data defined in the above sections will be saved in a specially formatted input file which has the following structure.

@T_CONNECTOR_DEFINITION {

@T_CONNECTOR_NAME { `TconName` **} {**

@CONNECTED_AT _A{ `WallAName`**}**
@CONNECTED_AT _B{ `WallBName`**}**
@CONNECTED_AT _C{ `WallCName`**}**
@NUMBER_OF_PASS_THROUGH_LAYERS{ `NbOfLayers`**}**

}

}

3.4.2 Examples

A few examples that describe the construction procedure of this type of section are shown below.

Example 1

This example shows a simple Tcon connector that connects 3 walls. All walls have 2 layers. This example also shows the axial strain field over the cross-section under the applied sectional loads.

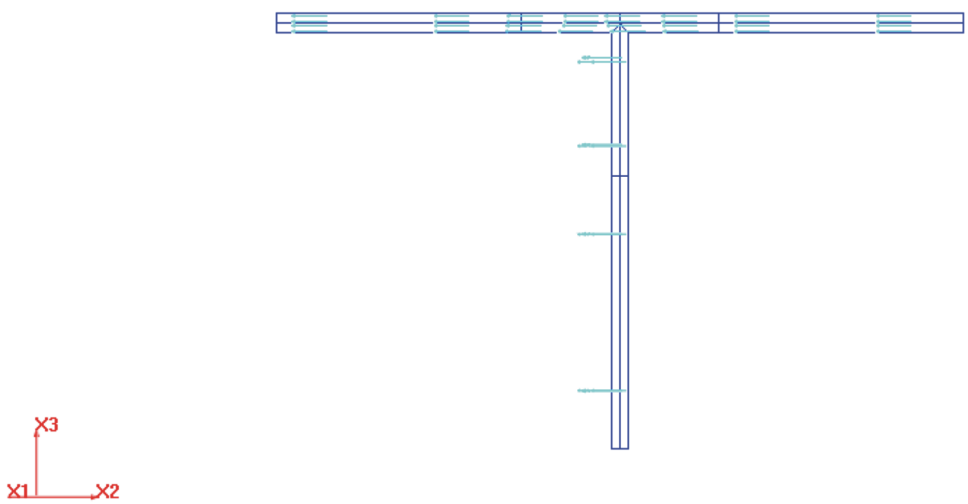

Figure 3.15: Example 1-Tcon connector with three Walls.

Example 2

This example shows the connection among 5 walls using 2 Tcon connectors. This example also shows the inverse of the reserve factor over the cross- section under the applied sectional loads.

Example 3

This example shows the connection among 8 walls using 6 Tcon connectors. All walls have more than 2 layers and ply-add/drop is also shown here. This example also shows the shear strain field over the cross-section under the applied sectional loads.

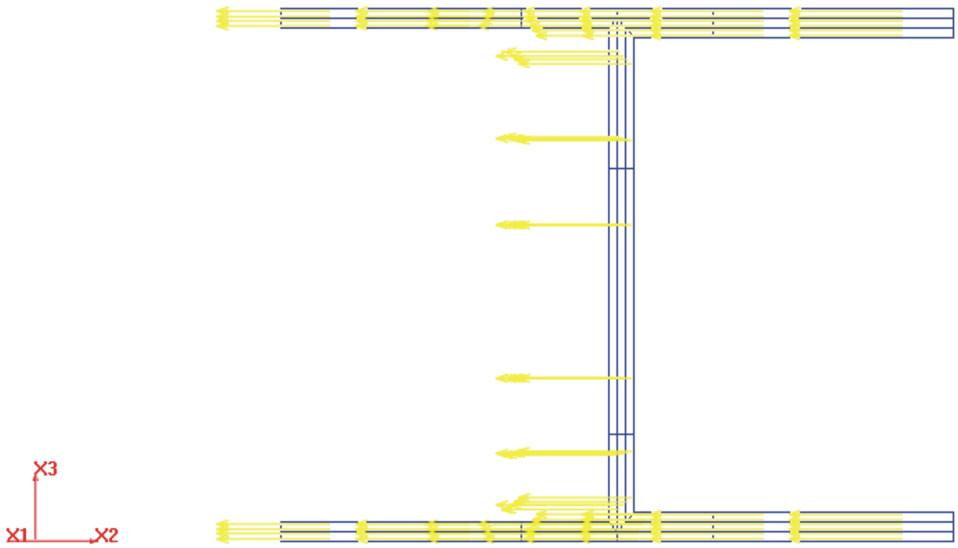

Figure 3.16: Example 2-two Tcon connectors with five Walls.

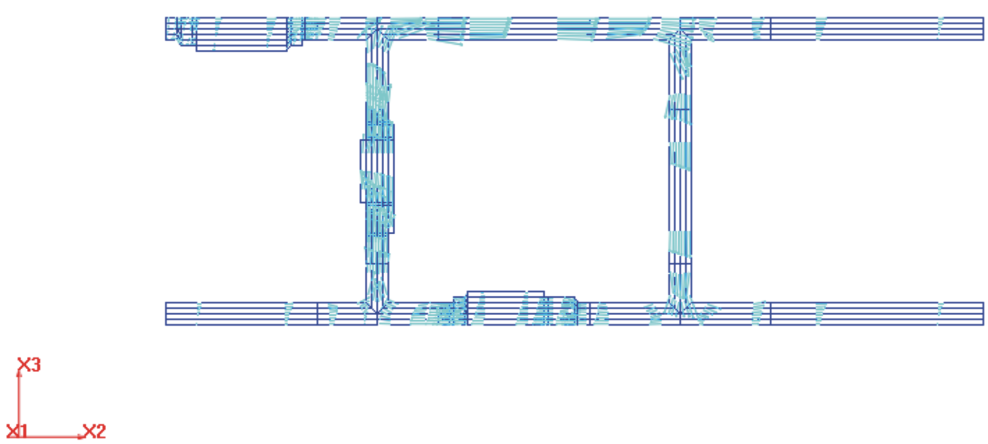

Figure 3.17: Example 3-Four Tcon connectors with Eight Walls.

3.5 Definition of Vcon sections

Vcon is a connector for connecting two walls like a V-shape. Two walls (Wall A and Wall B) connected with a Vcon, are depicted in fig. 3.18. Vcon has the same number of layers of walls at the junction. It is very useful for the trailing-edge construction of an airfoil section. User needs to provide the Wall A (connected at position A of Vcon) and Wall B(connected at position B of Vcon) information for constructing a Vcon.

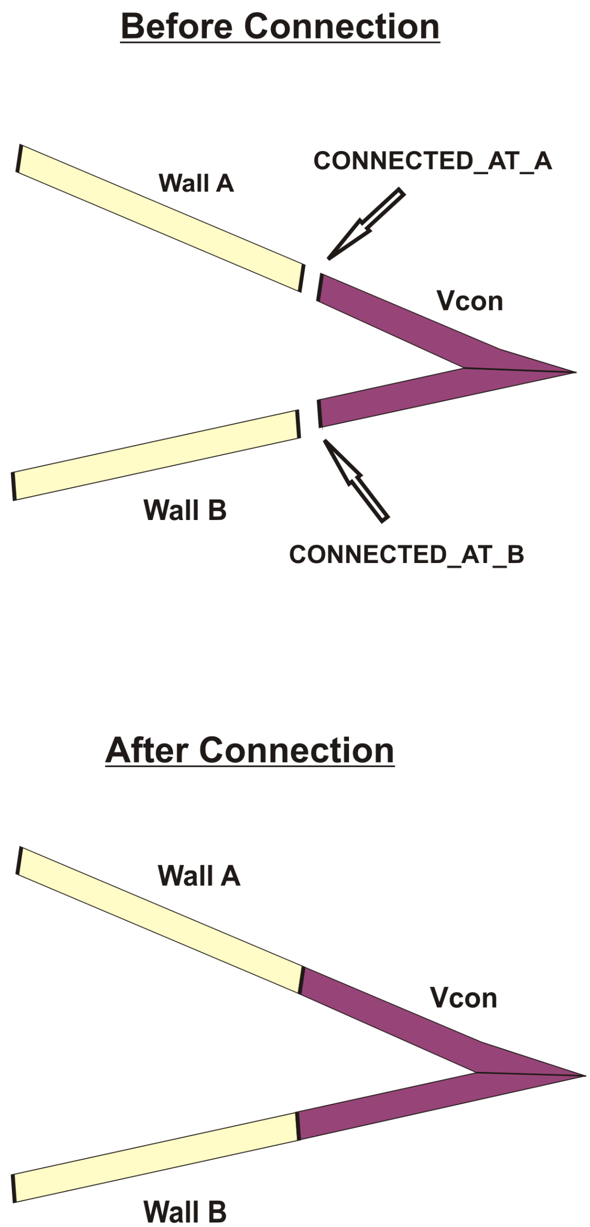

Figure 3.18: Two walls can be connected as above with a Vcon

The detail description of Vcon is shown in fig. 3.19. Basically wall information is used for Vcon construction. Points Pi and Ri (i= 1,2,3 .....n) are generated from WallA and WallB. Points Pi are the starting positions (U=0) of various layers of WallA. Similarly, points Ri are the final positions (at U=1) of the layers of WallB. Qi are the intersection points of the tangents of the layers of WallA (at U=0) and WallB (at U=1). Curves and surfaces are constructed from Pi, Qi and Ri. Solids are generated using the material and mesh properties. From WallA and WallB properties VconUp and VconLo sections are constructed.

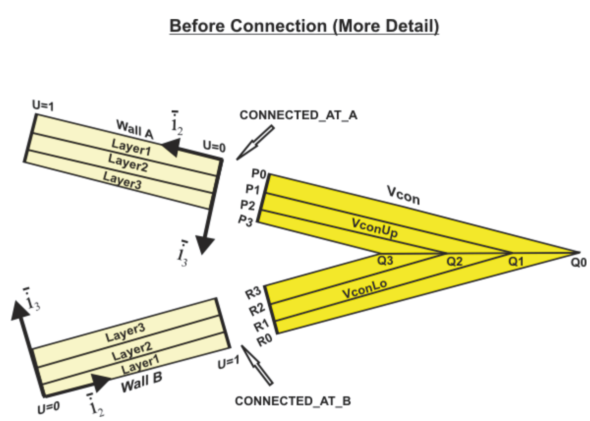

Figure 3.19: Two walls and Vcon (in detail)

3.5.1 Formatted input

The data defined in the above sections will be saved in a specially formatted input file which has the following structure.

@V_CONNECTOR_DEFINITION {

@V_CONNECTOR_NAME { `VconName` **} {**

@CONNECTED_AT _A{ `WallAName`**}**
@CONNECTED_AT _B{ `WallBName`**}**

}

}

3.5.2 Examples

An example that describes the construction procedure of this type of section is shown below.

Example 1

This example shows a Vcon connector that connects 2 walls. These walls have 2 layers. This example also shows the principal axes of inertia at the mass center.

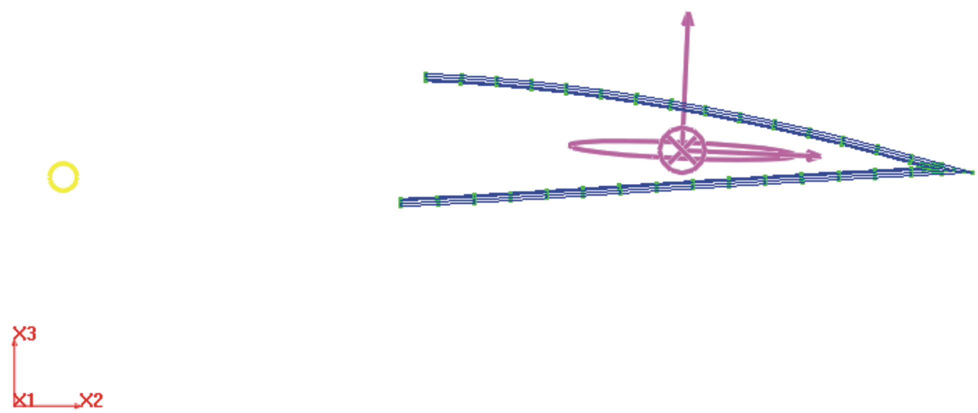

Figure 3.20: Example 1-Vcon connector with three Walls.

Chapter 4

Material properties

To fully define the cross-section of a beam, both its geometry and material properties must be defined. Material properties are defined through the buttons of the `Materials` menu item shown in fig. 4.1 and can be of the following type: definition of material properties as described in section 4.1, or definition of solid properties as described in section 4.2.

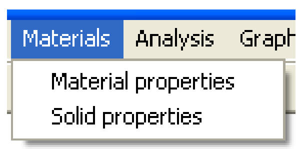

Figure 4.1: The `Materials` menu.

For the purpose of discretization, cross-sections are divided in a number of quadrangular areas, called "solids." For instance, consider the C-section shown in fig. 4.2; the section is divided into three solids labeled **TFlg**, **Web** and **BFlg**.

The material properties of each solid must be defined. If a solid is made of a single material, as depicted in the left portion of fig. 4.2, the physical properties of this material must be defined. This involve the definition of stiffness, mass, and strength characteristics of the material, as discussed in section 4.1. In other cases, the solid could be made of a stack of layered materials, as depicted in the right portion of fig. 4.2. This is a common occurrence when laminated composites materials are used. It is then becomes necessary to describe the thicknesses of the layers, the materials they are made of, and the fiber orientation angle. This more complex task is done by defining solid properties, as discussed in section 4.2.

When dealing with solid properties, a solid local axis system, $\mathcal{U} = (\bar{\imath}_1, \bar{u}, \bar{v})$, is defined. This local axis system is necessary to describe the configuration of the stack of layered materials. For instance, in the right portion of fig. 4.2, the layers of material run parallel to unit vector $\bar{u}$ of this local axis system. Furthermore, this local system will also be used to describe the fiber orientation angle when laminated composite materials are used.

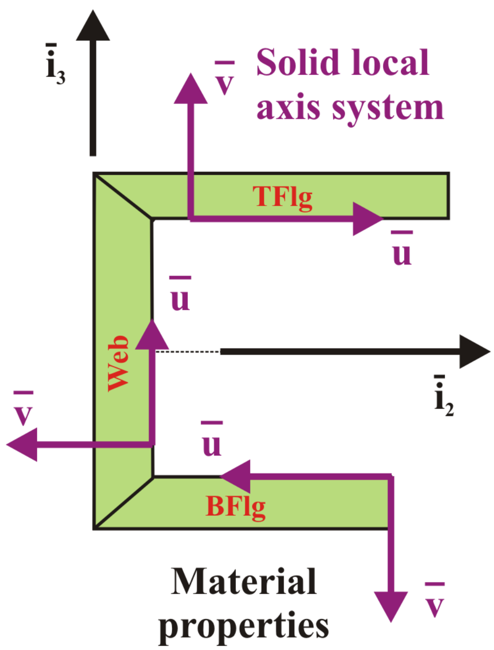

Figure 4.2: A C-section divided in three solids called **TFlg**, **Web** and **BFlg**. Left figure: the three solids are made of a single material. Right figure: the three solids are made of a stack of layered materials.

4.1 Definition of material properties

The definition of the physical properties of materials involve the definition of material density, material stiffness and strength characteristics, and the selection of a failure criterion. Materials to be defined can be of three distinct types: isotropic, orthotropic or transversely isotropic materials. These properties are defined by means of four dialog windows.

1. The *Material properties* dialog window as described in section 4.1.1, which defined the name of the materials, its mass density and type.

2. The *Stiffness* dialog window as described in section 4.1.2, which defines the stiffness coefficients for the material.

3. The *Failure Criterion* dialog window as described in section 4.1.3, which selects a failure criterion.

4. The *Strength* dialog window as described in section 4.1.4, which defines the strength characteristics of the material.

Material properties for a few standard materials, aluminum, titanium, and steel, are available in a template files. The material property names are `MatPropAluminum`, `MatPropTitanium`, `MatPropSteel`, for aluminum, titanium, and steel, respectively. To use these properties, invoke the *include command*, see section 8.1, to include one of the following files: C:\SectionBuilder\Materials\MATERIAL_SI.tpl, or C:\SectionBuilder\Materials\MATERIAL_US.tpl, or C:\SectionBuilder\Materials\MATERIAL_US_IN.tpl, for material properties in the SI, US, or US-IN unit systems, respectively.

4.1.1 The *Material properties* dialog window

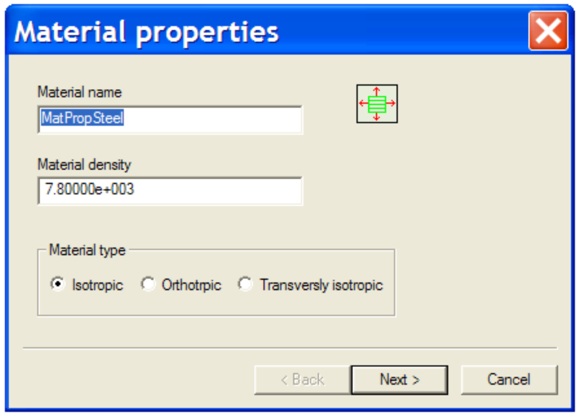

Figure 4.3: The `Material properties` dialog window.

The *Material properties* dialog window defines the following data for the material.

1. **Material name.** Enter a unique name for the material.
2. **Material density.** Enter the density of the material.
3. **Material type.** Materials will fall into three categories: isotropic, orthotropic and transversely isotropic materials [4, 5, 6]. Orthotropic materials possess two orthogonal planes of material property symmetry, implying the existence of a third as described in fig. 4.4; transversely isotropic materials feature a plane of material isotropy, *i.e.* properties are identical in all directions in this plane; finally, isotropic material have identical properties in all directions. In each case, a **material axis system**, $(\bar{e}_1, \bar{e}_2, \bar{e}_3)$, is defined that reflects the existence of various planes of symmetry and/or orthotropy.

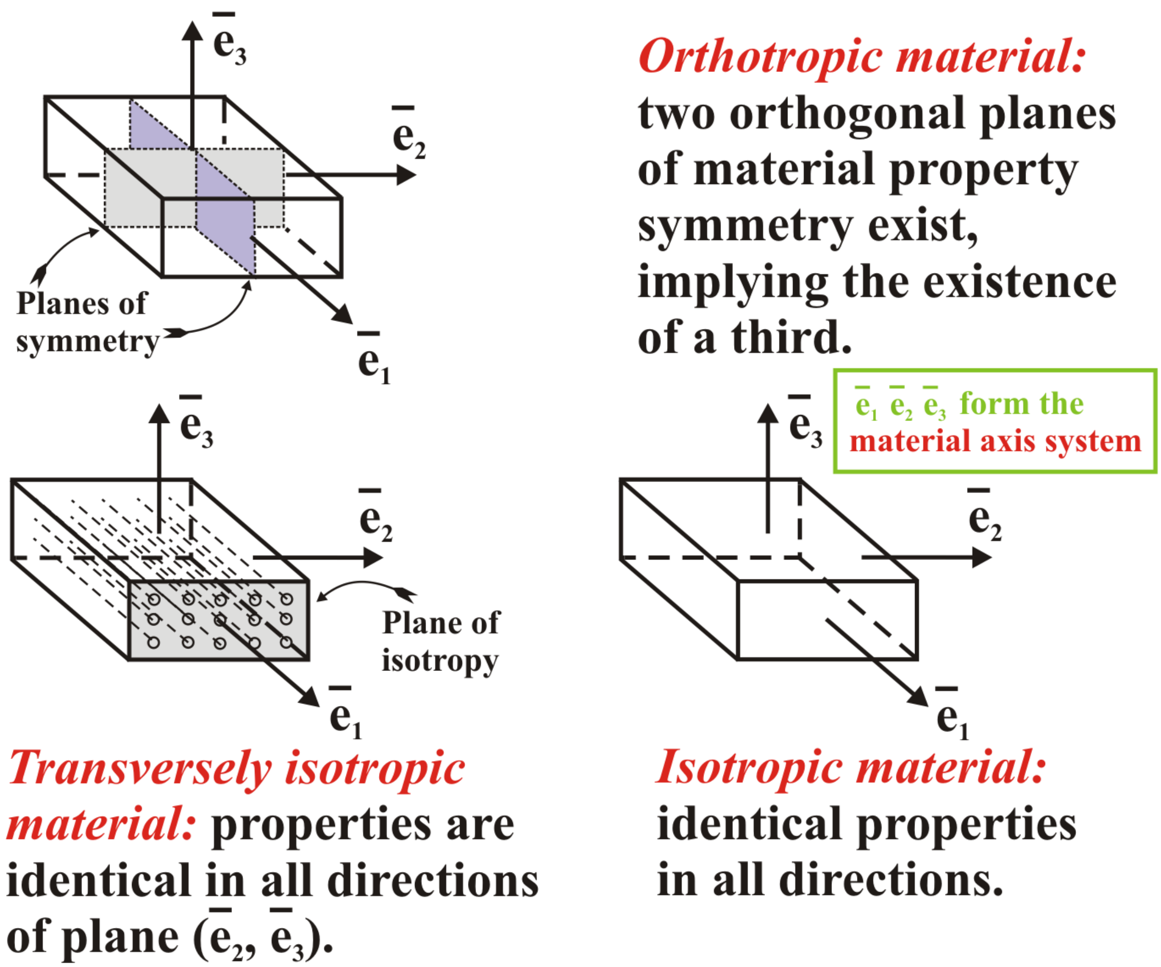

Figure 4.4: Orthotropic, transversely isotropic and isotropic materials.

4.1.2 The *Stiffness* dialog window

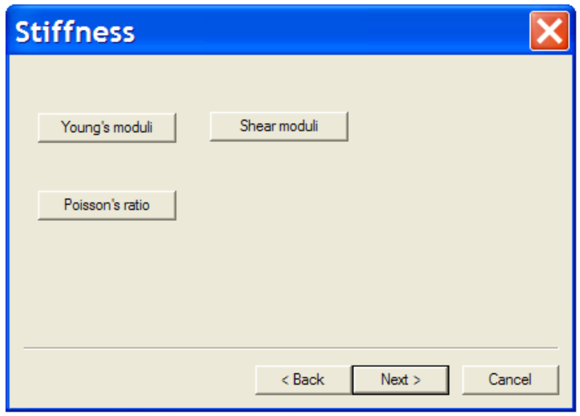

Figure 4.5: The *Stiffness* dialog window.

The *Stiffness* dialog window defines the stiffnesses of the material. For an orthotropic material, the compliance matrix [4, 5, 6] takes the following form

$$\left\{\begin{array}{c}\epsilon_1\\ \epsilon_2\\ \epsilon_3\\ \gamma_{23}\\ \gamma_{13}\\ \gamma_{12}\end{array}\right\} = \begin{bmatrix} 1/E_1 & -\nu_{12}/E_1 & -\nu_{13}/E_1 & 0 & 0 & 0\\ -\nu_{12}/E_1 & 1/E_2 & -\nu_{23}/E_2 & 0 & 0 & 0\\ -\nu_{13}/E_1 & -\nu_{23}/E_2 & 1/E_3 & 0 & 0 & 0\\ 0 & 0 & 0 & 1/G_{23} & 0 & 0\\ 0 & 0 & 0 & 0 & 1/G_{13} & 0\\ 0 & 0 & 0 & 0 & 0 & 1/G_{12}\end{bmatrix} \left\{\begin{array}{c}\sigma_1\\ \sigma_2\\ \sigma_3\\ \tau_{23}\\ \tau_{13}\\ \tau_{12}\end{array}\right\}. \tag{4.1}$$

The array of *strain components* is

$$\underline{\epsilon}^T = \lfloor \epsilon_1, \epsilon_2, \epsilon_3, \gamma_{23}, \gamma_{13}, \gamma_{12} \rfloor, \tag{4.2}$$

where ϵ_1, ϵ_2 and ϵ_3 are the axial strains components along unit vectors $\bar{e}_1$, $\bar{e}_2$ and $\bar{e}_3$, respectively. The orthonormal units vectors $\bar{e}_1$, $\bar{e}_2$ and $\bar{e}_3$ form the **material axis system**, as illustrated in fig. 4.4; the corresponding engineering shear strains components are γ_{23}, γ_{13} and γ_{12}. The array of *stress components* is

$$\underline{\sigma}^T = \lfloor \sigma_1, \sigma_2, \sigma_3, \tau_{23}, \tau_{13}, \tau_{12} \rfloor, \tag{4.3}$$

where σ_1, σ_2 and σ_3 are the axial stress components along unit vectors $\bar{e}_1$, $\bar{e}_2$ and $\bar{e}_3$, respectively; the corresponding shearing stresses are τ_{23}, τ_{13} and τ_{12}.

In view of eq. (4.1), the material stiffness properties are characterized by three distinct **Young's moduli**: E_1, E_2 and E_3 along unit vectors $\bar{e}_1$, $\bar{e}_2$ and $\bar{e}_3$, respectively; three **Poisson's ratios**: ν_{12}, ν_{13} and ν_{23}; and three **shearing moduli**: G_{12}, G_{13} and G_{23}.

1. For an *orthotropic material*, the following *nine* properties are required.
   (a) **Young's moduli:** E_1, E_2 and E_3.
   (b) **Shear moduli:** G_{12}, G_{13} and G_{23}.
   (c) **Poisson's ratios:** ν_{12}, ν_{13} and ν_{23}.
2. For a *transversely isotropic material*, the following *five* properties are required
   (a) **Young's moduli:** E_1 and E_2.
   (b) **Shear moduli:** G_{12}.
   (c) **Poisson's ratios:** ν_{12} and ν_{23}.

In this case, $E_3 = E_2$, $G_{13} = G_{12}$ and $\nu_{13} = \nu_{12}$: in view of the isotropy in the $(\bar{e}_2, \bar{e}_3)$ plane, the subscripts $(.)_2$ and $(.)_3$ can be interchanged. Furthermore, the isotropy of plane $(\bar{e}_2, \bar{e}_3)$ implies $G_{23} = E_2/(2(1 + \nu_{23}))$.

3. For an *isotropic material*, the following *two* properties are required

   (a) **Young's modulus:** E.

   (b) **Poisson's ratios:** ν.

   In this case, the isotropy of the material implies $E_1 = E_2 = E_3 = E$, $\nu_{12} = \nu_{13} = \nu_{23} = \nu$, and $G_{12} = G_{13} = G_{23} = E/(2(1 + \nu))$.

4.1.3 The *Failure Criterion* dialog window

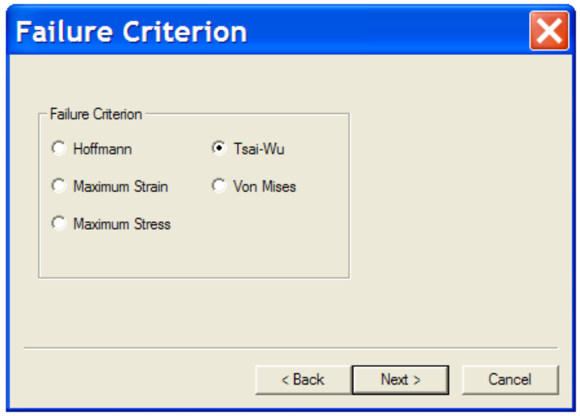

Figure 4.6: The *Failure Criterion* dialog window.

The *Failure Criterion* dialog window defines the failure criterion to be used for this material type. The failure criterion can be selected from the following list.

1. *Hoffmann Criterion [4].* This criterion is used for *transversely isotropic material only.* At failure, the following equation is satisfied

$$s_1^2 - F_{12}s_1s_2 + s_2^2 + s_6^2 + F_1s_1 + F_2s_2 = 1, \tag{4.4}$$

where $s_1 = \sigma_1/(\sigma_1^{\rm aT}\sigma_1^{\rm aC})^{1/2}$, $s_2 = \sigma_2/(\sigma_2^{\rm aT}\sigma_2^{\rm aC})^{1/2}$ and $s_6 = \tau_{12}/\tau_{12}^{\rm a}$, σ_1 and σ_2 are the **stresses along the material axes** $\bar{e}_1$ and $\bar{e}_2$, respectively, and τ_{12} the corresponding shear stress. The allowable tensile and compressive stresses along axis $\bar{e}_1$ are denoted $\sigma_1^{\rm aT}$ and $\sigma_1^{\rm aC}$, respectively. Similarly, the allowable tensile and compressive stresses along axis $\bar{e}_2$ are denoted $\sigma_2^{\rm aT}$ and $\sigma_2^{\rm aC}$, respectively. Finally, the allowable shear stresses in plane ($\bar{e}_1$, $\bar{e}_2$) is denoted $\tau_{12}^{\rm a}$. All these quantities are defined in section 4.1.4. The two coefficients F_1 and F_2 are

$$F_1 = (\sigma_1^{\rm aC} - \sigma_1^{\rm aT})/(\sigma_1^{\rm aT}\sigma_1^{\rm aC})^{1/2}, \tag{4.5}$$

and

$$F_2 = (\sigma_2^{\rm aC} - \sigma_2^{\rm aT})/(\sigma_2^{\rm aT}\sigma_2^{\rm aC})^{1/2}. \tag{4.6}$$

The last coefficient is

$$F_{12} = (\sigma_2^{\rm aT}\sigma_2^{\rm aC})^{1/2}/(\sigma_1^{\rm aT}\sigma_1^{\rm aC})^{1/2}. \tag{4.7}$$

2. *Maximum Strain Criterion [4].* This criterion is used for *transversely isotropic material only.* At failure, one of the following equations is satisfied.

$$\epsilon_1 = \left\{ \begin{array}{lll} \sigma_1^{\rm aT}/E_1 & \text{if} & \epsilon_1 > 0 \\ \sigma_1^{\rm aC}/E_1 & \text{if} & \epsilon_1 < 0 \end{array} \right., \tag{4.8}$$

$$\epsilon_2 = \left\{ \begin{array}{lll} \sigma_2^{\rm aT}/E_2 & \text{if} & \epsilon_2 > 0 \\ \sigma_2^{\rm aC}/E_2 & \text{if} & \epsilon_2 < 0 \end{array} \right., \tag{4.9}$$

$$\gamma_{12} = \tau_{12}^{\rm a}/G_{12}, \tag{4.10}$$

where ϵ_1 and ϵ_2 are the **strains along the material axes** $\bar{e}_1$ and $\bar{e}_2$, respectively, and γ_{12} the corresponding shear strain. The allowable tensile and compressive stresses along axis $\bar{e}_1$ are denoted $\sigma_1^{\rm aT}$ and $\sigma_1^{\rm aC}$, respectively. Similarly, the allowable tensile and compressive stresses along axis $\bar{e}_2$ are denoted $\sigma_2^{\rm aT}$ and $\sigma_2^{\rm aC}$, respectively. Finally, the allowable shear stresses in plane ($\bar{e}_1$, $\bar{e}_2$) is denoted $\tau_{12}^{\rm a}$. All these quantities are defined in section 4.1.4.

3. *Maximum Stress Criterion [4].* This criterion is used for *transversely isotropic material only.* At failure, one of the following equations is satisfied.

$$\sigma_1 = \begin{cases} \sigma_1^{\mathrm{aT}} & \text{if} \quad \sigma_1 > 0 \\ \sigma_1^{\mathrm{aC}} & \text{if} \quad \sigma_1 < 0 \end{cases}, \tag{4.11}$$

$$\sigma_2 = \begin{cases} \sigma_2^{\mathrm{aT}} & \text{if} \quad \sigma_2 > 0 \\ \sigma_2^{\mathrm{aC}} & \text{if} \quad \sigma_2 < 0 \end{cases}, \tag{4.12}$$

$$\tau_{12} = \tau_{12}^{\mathrm{a}}, \tag{4.13}$$

where σ_1 and σ_2 are the **stresses along the material axes** $\bar{e}_1$ and $\bar{e}_2$, respectively, and τ_{12} the corresponding shear stress. The allowable tensile and compressive stresses along axis $\bar{e}_1$ are denoted σ_1^{aT} and σ_1^{aC}, respectively. Similarly, the allowable tensile and compressive stresses along axis $\bar{e}_2$ are denoted σ_2^{aT} and σ_2^{aC}, respectively. Finally, the allowable shear stresses in plane ($\bar{e}_1$, $\bar{e}_2$) is denoted τ_{12}^{a}. All these quantities are defined in section 4.1.4.

4. *Tsai-Wu Criterion [4].* This criterion is used for *transversely isotropic material only.* At failure, the following equation is satisfied

$$s_1^2 - s_1 s_2 + s_2^2 + s_6^2 + F_1 s_1 + F_2 s_2 = 1,$$

where $s_1 = \sigma_1/(\sigma_1^{\mathrm{aT}}\sigma_1^{\mathrm{aC}})^{1/2}$, $s_2 = \sigma_2/(\sigma_2^{\mathrm{aT}}\sigma_2^{\mathrm{aC}})^{1/2}$ and $s_6 = \tau_{12}/\tau_{12}^{\mathrm{a}}$, σ_1 and σ_2 are the **stresses along the material axes** $\bar{e}_1$ and $\bar{e}_2$, respectively, and τ_{12} the corresponding shear stress. The allowable tensile and compressive stresses along axis $\bar{e}_1$ are denoted σ_1^{aT} and σ_1^{aC}, respectively. Similarly, the allowable tensile and compressive stresses along axis $\bar{e}_2$ are denoted σ_2^{aT} and σ_2^{aC}, respectively. Finally, the allowable shear stresses in plane ($\bar{e}_1$, $\bar{e}_2$) is denoted τ_{12}^{a}. All these quantities are defined in section 4.1.4. The two coefficients F_1 and F_2 are

$$F_1 = (\sigma_1^{\mathrm{aC}} - \sigma_1^{\mathrm{aT}})/(\sigma_1^{\mathrm{aT}}\sigma_1^{\mathrm{aC}})^{1/2},$$

and

$$F_2 = (\sigma_2^{\mathrm{aC}} - \sigma_2^{\mathrm{aT}})/(\sigma_2^{\mathrm{aT}}\sigma_2^{\mathrm{aC}})^{1/2}.$$

5. *Von Mises Criterion.* This criterion is used for *isotropic materials only.* At failure, the following equation is satisfied

$$\frac{1}{2}\left[(s_1 - s_2)^2 + (s_2 - s_3)^2 + (s_3 - s_1)^2\right] = 1,$$

where $s_1 = \sigma_1/\sigma^{\mathrm{aT}}$, $s_2 = \sigma_2/\sigma^{\mathrm{aT}}$ and $s_3 = \sigma_3/\sigma^{\mathrm{aT}}$, σ_1, σ_2 and σ_3 are **the principal stresses**, and σ^{aT} the allowable stress in tension defined in section 4.1.4. Note that since the material is assumed to be isotropic, its strength is identical in all directions; furthermore, its compressive and tensile strengths are assumed to be identical.

4.1.4 The *Strength* dialog window

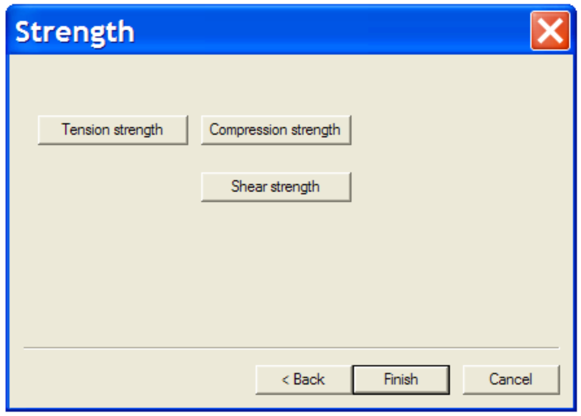

Figure 4.7: The *Strength* dialog window.

The *Strength* dialog window defines the strength properties of the material. In general, nine different strength values can be defined

1. *Allowable stress in tension:* σ_1^{aT}, σ_2^{aT} and σ_3^{aT}, along the material axes $\bar{e}_1$, $\bar{e}_2$ and $\bar{e}_3$, respectively.
2. *Allowable stress in compression:* σ_1^{aC}, σ_2^{aC} and σ_3^{aC}, along the material axes $\bar{e}_1$, $\bar{e}_2$ and $\bar{e}_3$, respectively.
3. *Allowable shear stress:* τ_{12}^{a}, τ_{13}^{a} and τ_{23}^{a}.

1. For an *orthotropic material*, all *nine* strength properties are required
   (a) **Tension strength:** σ_1^{aT}, σ_2^{aT} and σ_3^{aT}.
   (b) **Compression strength:** σ_1^{aC}, σ_2^{aC} and σ_3^{aC}.
   (c) **Shear strength:** τ_{12}^{a}, τ_{13}^{a} and τ_{23}^{a}.
2. For a *transversely isotropic material*, the following *five* properties are required
   (a) **Tension strength:** σ_1^{aT} and σ_2^{aT}.
   (b) **Compression strength:** σ_1^{aC} and σ_2^{aC}.
   (c) **Shear strength:** τ_{12}^{a}.

   In this case, $\sigma_3^{\mathrm{aT}} = \sigma_2^{\mathrm{aT}}$, $\sigma_3^{\mathrm{aC}} = \sigma_2^{\mathrm{aC}}$ and $\tau_{13}^{\mathrm{a}} = \tau_{12}^{\mathrm{a}}$: in view of the isotropy in the $(\bar{e}_2, \bar{e}_3)$ plane, the subscripts $(.)_2$ and $(.)_3$ can be interchanged. Furthermore, the isotropy of plane $(\bar{e}_2, \bar{e}_3)$ implies $\tau_{23}^{\mathrm{a}} = \sigma_2^{\mathrm{aT}}/\sqrt{3}$.
3. For an *isotropic material*, a *single* property is required
   (a) **Tension strength:** σ^{a}.

   In this case, $\sigma_1^{\mathrm{aT}} = \sigma_2^{\mathrm{aT}} = \sigma_3^{\mathrm{aT}} = \sigma^{\mathrm{a}}$, $\sigma_1^{\mathrm{aC}} = \sigma_2^{\mathrm{aC}} = \sigma_3^{\mathrm{aC}} = \sigma^{\mathrm{a}}$ and $\tau_{12}^{\mathrm{a}} = \tau_{13}^{\mathrm{a}} = \tau_{23}^{\mathrm{a}} = \sigma^{\mathrm{a}}/\sqrt{3}$.

4.1.5 Formatted input

The data that defines materials properties as described in the above section will be saved in a specially formatted input file which has the following structure.

@MATERIAL_PROPERTY_DEFINITION {

@MATERIAL_PROPERTY_DEFINITION { `MaterialName` **} {**

@MATERIAL_PROPERTY_TYPE { `MaterialType` **}**
@FAILURE_CRITERION_TYPE { `FailureCriterion` **}**
@MATERIAL_DENSITY { ρ **}**
@YOUNG'S_MODULUS { E_1, E_2, E_3 **}**
@POISSON'S_RATIO { $\nu_{12}, \nu_{13}, \nu_{23}$ **}**
@SHEAR_MODULUS { G_{12}, G_{13}, G_{23} **}**
@ALLOWABLE_TENSILE_STRESS { $\sigma_1^{\mathrm{aT}}, \sigma_2^{\mathrm{aT}}, \sigma_3^{\mathrm{aT}}$ **}**
@ALLOWABLE_COMPRESSIVE_STRESS { $\sigma_1^{\mathrm{aC}}, \sigma_2^{\mathrm{aC}}, \sigma_3^{\mathrm{aC}}$ **}**
@ALLOWABLE_SHEAR_STRESS { $\tau_{12}^{\mathrm{a}}, \tau_{13}^{\mathrm{a}}, \tau_{23}^{\mathrm{a}}$ **}**

}

}

4.2 Definition of solid properties

For the purpose of discretization, cross-sections are divided in a number of quadrangular areas, called "solids." For instance, consider the C-section shown in fig. 2.22; the section is divided into three zones: *(1)* the top reinforcement flange consisting of a single solid labeled *TFlgR*, *(2)* the C-section it self consisting of the solids labeled **TFlg**, **Web**, **BFlg**, and *(3)* the bottom reinforcement flange consisting of a single solid labeled *BFlgR*. The physical properties of each solid must be defined. If a solid is made of a single material, the properties of this material can be directly defined as material properties as described in section 4.1. In other cases, the solid consists of layered materials, such as laminated composites; it is then becomes necessary to use the present solid properties that are defined by means of two dialog windows.

1. The *Solid material properties* dialog window as described in section 4.2.1, which defines the name of the solid properties and the axis system to be used.

2. The *layer List* dialog window, section 4.2.2, which defines the thickness, orientation angle, and material for each of the layers.

4.2.1 The *Solid material properties* dialog window

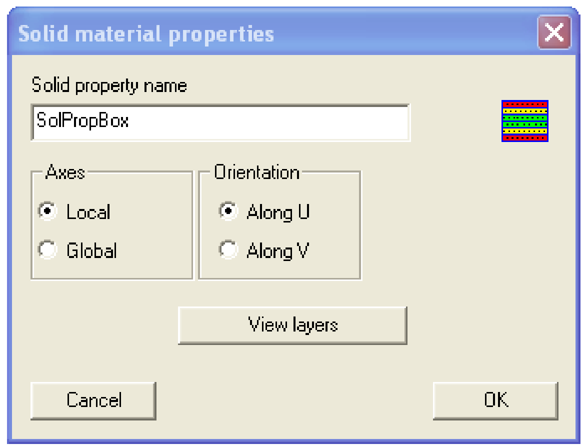

Figure 4.8: The *Solid material properties* dialog window.

The *Solid material properties* dialog window as described in fig. 4.8, defines the following data for the solid properties.

1. **Solid properties name.** Enter a unique name for the solid properties.
2. **Axes Flag.** This flag can be set to `local` or `global`. This option will play an important role in the definition of the material axes orientation as described in below.
3. **Layer orientation.** Layers can be selected to be oriented `Along U` or `Along V`.

Fig. 4.9 depicts a typical solid in the cross-section with the **local axis system**, $\mathcal{U} = (\bar{\imath}_1, \bar{u}, \bar{v})$. The layers of material can run along the $\bar{u}$ or $\bar{v}$ axes, as illustrated in the left and right portions of the figure, respectively.

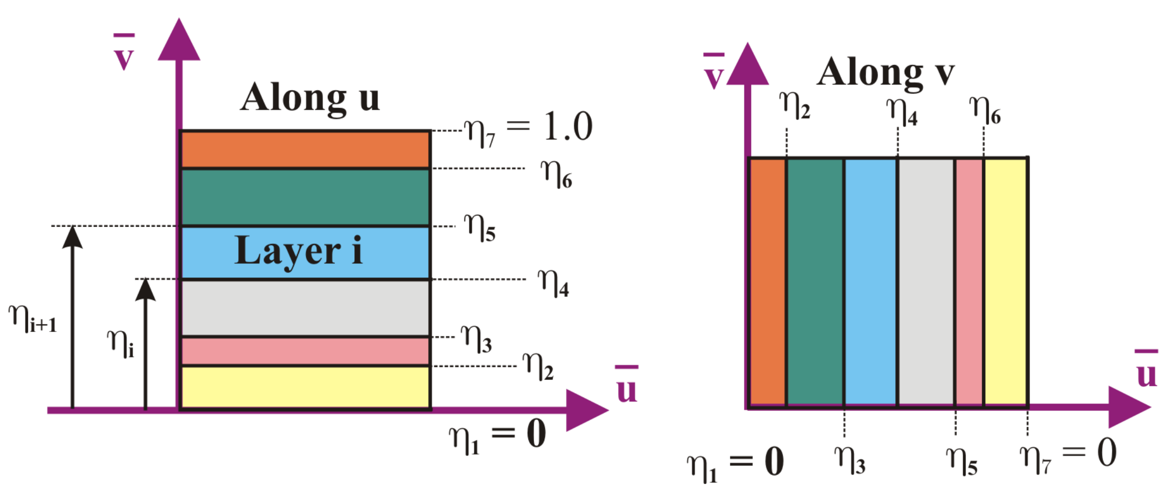

Figure 4.9: Stacks of layers defining solid properties. In the left portion of the figure, the layers all run parallel to the $\bar{u}$ axis; this corresponds to the `Along U` option. In the right portion of the figure, the layers all run parallel to the $\bar{v}$ axis; this corresponds to the `Along V` option.

4.2.2 The *Layer List* dialog window

Clicking `View Layers` button of the *Name* dialog tab as described in fig. 4.8, opens the *Layer List* dialog window shown in fig. 4.10. As shown in fig. 4.9, solid properties define a stack of layers, each layer has its own thickness, orientation angles and material. The thickness of the layer is determined by non-dimensional coordinates η_i and η_{i+1}. Referring to fig. 4.9, `Layer i` extends from coordinate η_i to coordinate η_{i+1}. If the stack features `n` layers, a total of `n+1` entries must appear in the layer list: the list **must start** with coordinate $\eta_1 = 0.0$ and **must end** with coordinate $\eta_{n+1} = 1.0$; the intermediate coordinates **must appear in ascending order**. The layer list shown in fig. 4.10 contains the following information.

1. The first column of the layer list is the layer index, i.
2. The second column gives the starting location coordinate, η_i.
3. The next two columns give the material coordinate system orientation angles, α and β.
4. The last column gives the name of the material properties as described in section 4.1, the layer is made of.

The orientation angles and material properties appearing in the last line of the layer list repeat the entries on the previous line.

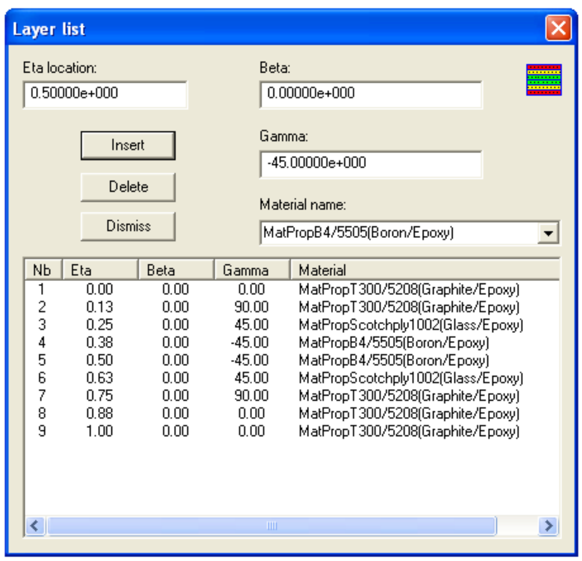

Figure 4.10: The *Layer List* dialog window. The layer list feature 9 entries for this 8 layer stack.

The layer orientation angles (*Axes Flag* set to "Local")

The two orientation angles, β and γ define the orientation of the **material axis system**, $\mathcal{E} = (\bar{e}_1, \bar{e}_2, \bar{e}_3)$ with respect to the global reference frame, $\mathcal{I} = (\bar{\imath}_1, \bar{\imath}_2, \bar{\imath}_3)$. Material properties as described in section 4.1, are defined in the material axis system as described in fig. 4.4.

A sequence of *three planar rotations* brings the global reference frame, $\mathcal{I}$, to the material axis system, $\mathcal{E}$, as illustrated in fig. 4.11.

1. The *first planar rotation* is of magnitude α about axis $\bar{\imath}_1$ and brings the reference axis system, $\mathcal{I} = (\bar{\imath}_1, \bar{\imath}_2, \bar{\imath}_3)$, to the solid local axis system, $\mathcal{U} = (\bar{\imath}_1, \bar{u}, \bar{v})$. Angle α is determined by the geometry of the section.

2. The *second planar rotation* is of magnitude β about axis $\bar{\imath}_1$ and brings the solid local axis system, $\mathcal{U} = (\bar{\imath}_1, \bar{u}, \bar{v})$, to frame $\mathcal{B} = (\bar{\imath}_1, \bar{b}_2, \bar{b}_3)$. Angle β is defined in the layer list. Since these first two planar rotations take place about the same axis $\bar{\imath}_1$, they can be combined into a single planar rotation of magnitude $(\alpha + \beta)$ about axis $\bar{\imath}_1$.

3. The *third planar rotation* is of magnitude γ about axis $\bar{b}_3$ and brings frame $\mathcal{B} = (\bar{\imath}_1, \bar{b}_2, \bar{b}_3)$, to the material axis system, $\mathcal{E} = (\bar{e}_1, \bar{e}_2, \bar{e}_3)$. Angle γ is defined in the layer list.

Note that positive angles β and γ correspond to positive rotations about axes $\bar{\imath}_1$ and $\bar{b}_3$, respectively, following the right hand rule. If the layer is a transversely isotropic material such as a unidirectional layer of composite, angle $\beta = 0$ and angle γ corresponds to the fiber orientation angle.

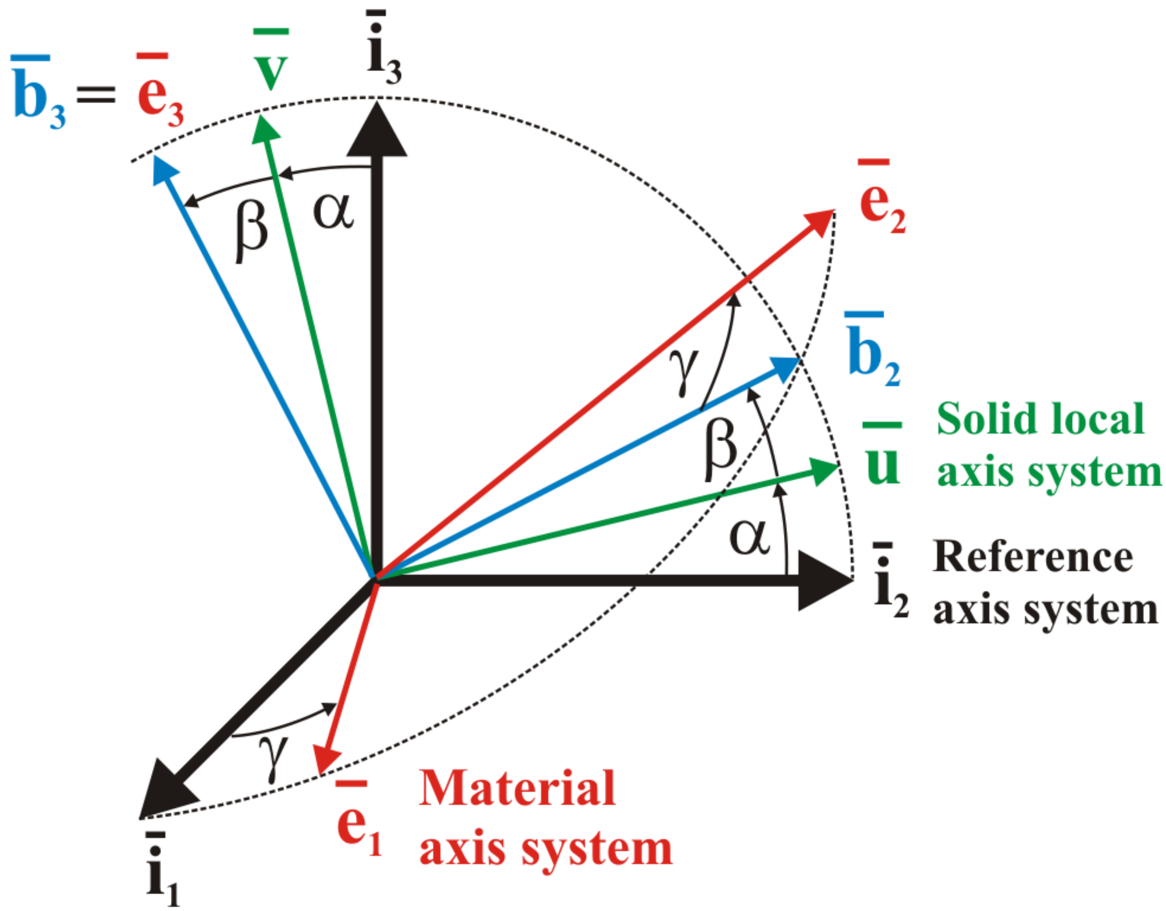

Figure 4.11: Orientation of the material axis system using the **Local Axes** option.

The layer orientation angles (*Axes Flag* set to "Global")

The two orientation angles, β and γ define the orientation of the **material axis system**, $\mathcal{E} = (\bar{e}_1, \bar{e}_2, \bar{e}_3)$ with respect to the reference axis system, $\mathcal{I} = (\bar{\imath}_1, \bar{\imath}_2, \bar{\imath}_3)$. Material properties as described in section 4.1, are defined in the material axis system as described in fig. 4.4.

A sequence of *two planar rotations* brings the global reference frame, $\mathcal{I}$, to the material axis system, $\mathcal{E}$, as illustrated in fig. 4.12.

1. The *first planar rotation* is of magnitude β about axis $\bar{\imath}_1$ and brings the reference axis system, $\mathcal{I} = (\bar{\imath}_1, \bar{\imath}_2, \bar{\imath}_3)$, to frame $\mathcal{B} = (\bar{\imath}_1, \bar{b}_2, \bar{b}_3)$. Angle β is defined in the layer list.

2. The *second planar rotation* is of magnitude γ about axis $\bar{b}_3$ and brings frame $\mathcal{B} = (\bar{\imath}_1, \bar{b}_2, \bar{b}_3)$, to the material axis system, $\mathcal{E} = (\bar{e}_1, \bar{e}_2, \bar{e}_3)$.

Note that positive angles β and γ correspond to positive rotations about axes $\bar{\imath}_1$ and $\bar{b}_3$, respectively, following the right hand rule. It is important to note that in this scheme, while the layer orientation depends on the local axis system, the determination of the material axis system orientation is independent of that of the local system.

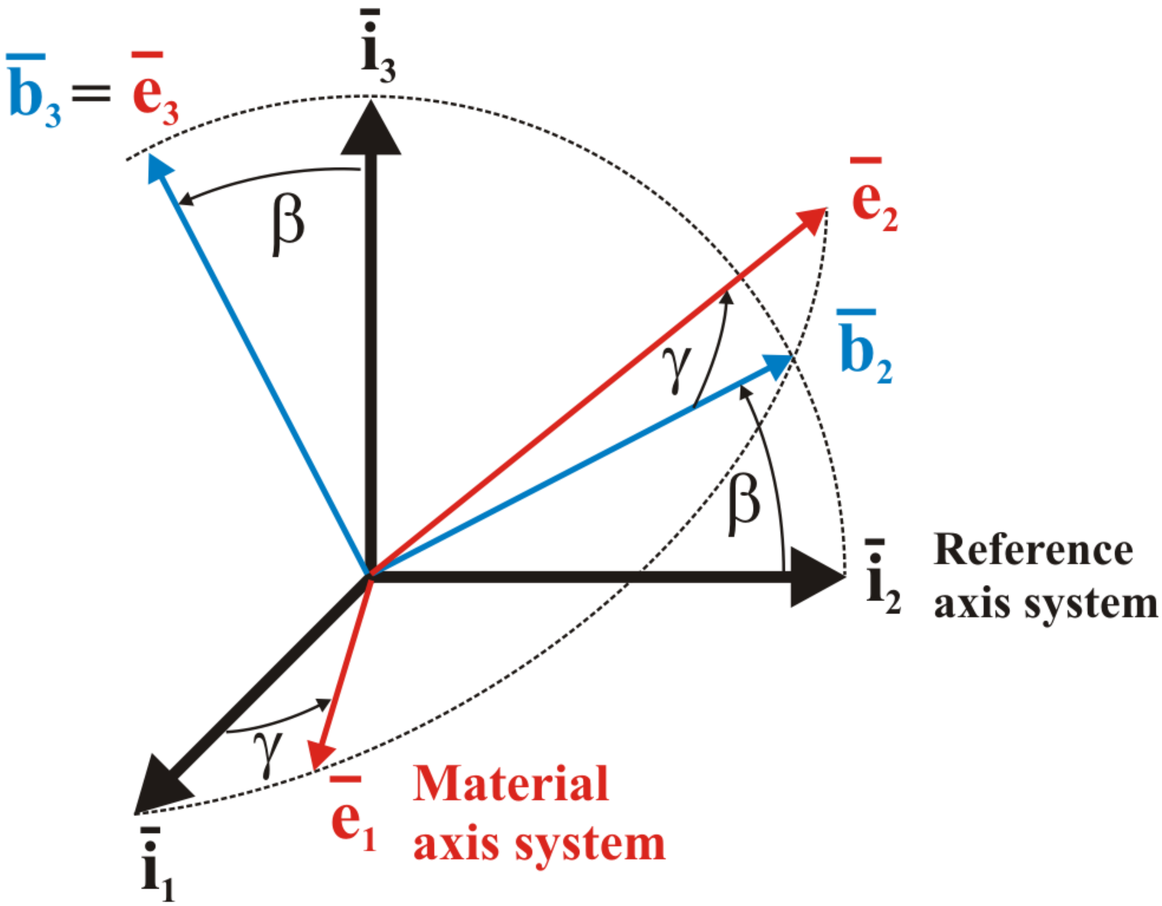

Figure 4.12: Orientation of the material axis system using the **Global Axes** option.

Chapter 5

Mesh density

Mesh density is an important parameter that will influence the quality of the finite element computation, and consequently, the accuracy of the predicted sectional stiffness characteristics and stress distributions. Mesh density is controlled by an integer parameter, m. A characteristic overall dimension, D, of the section is estimated first; next, a characteristic finite element dimension is computed as $d = D/m$. The mesh process then attempts to create finite elements that are approximately of size d. The meshing process recognizes the potential presence of layered materials: each layer is meshed independently to avoid smearing of the material properties.

5.1 Example

Consider the C-section shown in fig. 5.1. Mesh density parameters, $m = 4$, 8 and 16 used to create the meshes illustrated in the figure. Note that for $m = 4$ and 8 a single finite element is used through the thickness of the wall. For higher values of m, more than two or more finite elements are used through the thickness of the wall.

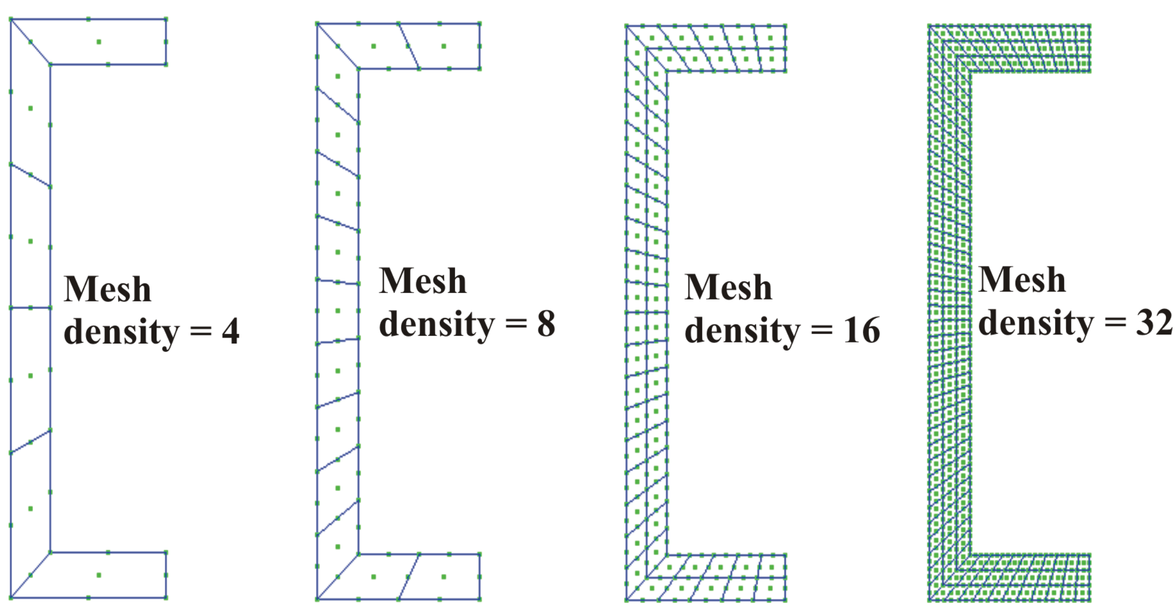

Figure 5.1: Mesh density variation effect on the C-section

Chapter 6

Applied loading

6.1 Sectional loads

External loads applied to the cross-section are defined defined by means of a load vector and of a moment vector. A *loading condition* is defined by a single dialog window, the `Sectional loads` dialog window.

6.1.1 The *Sectional loads* dialog window

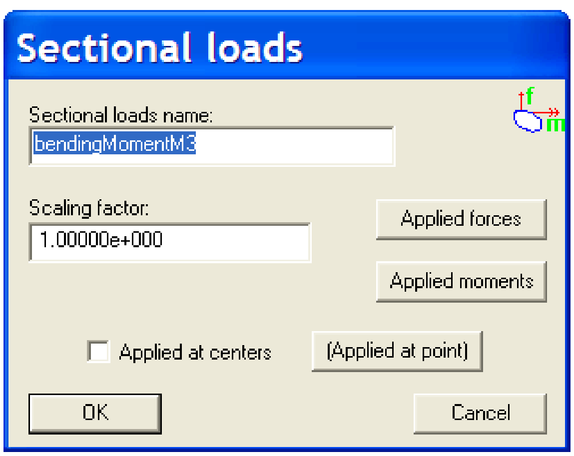

Figure 6.1: The *Sectional loads* dialog window.

The *Sectional loads* dialog window defines the *loading condition* with the following data

1. **Sectional loads name.** Enter a unique name for the *loading condition*.
2. **Scaling factor.** This scaling factor is a multiplicative factor that will affect both the force and the moment vectors.
3. **Applied Forces.** The `Applied Forces` button allows the definition of the three components of the externally applied force vector. As shown in fig. 6.2, the three components of force N_1, V_2 and V_3 are acting **along** the axes $\bar{\imath}_1$, $\bar{\imath}_2$ and $\bar{\imath}_3$, respectively. These forces are applied **at the origin of the axis system.**
4. **Applied Moments.** The `Applied Moment` button allows the definition of the three components of the externally applied moment vector. As shown in fig. 6.2, the three components of moment M_1, M_2 and M_3 are acting **about** the axes $\bar{\imath}_1$, $\bar{\imath}_2$ and $\bar{\imath}_3$, respectively.

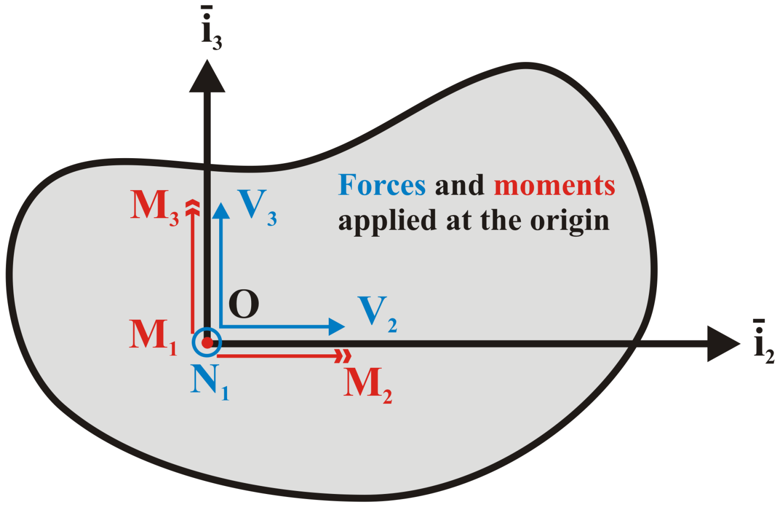

Figure 6.2: Externally applied force and moment components acting on the cross-section.

5. **Applied at Centers.** The flag `Applied at Centers` affect the way in which the previously defined forces and moments are applied to the cross-section. If the box `Applied at centers` is checked, the axial force N_1 is **applied at the centroid** and the transverse shear forces V_2 and V_3 are **applied at the shear center**, as depicted in fig. 6.3.

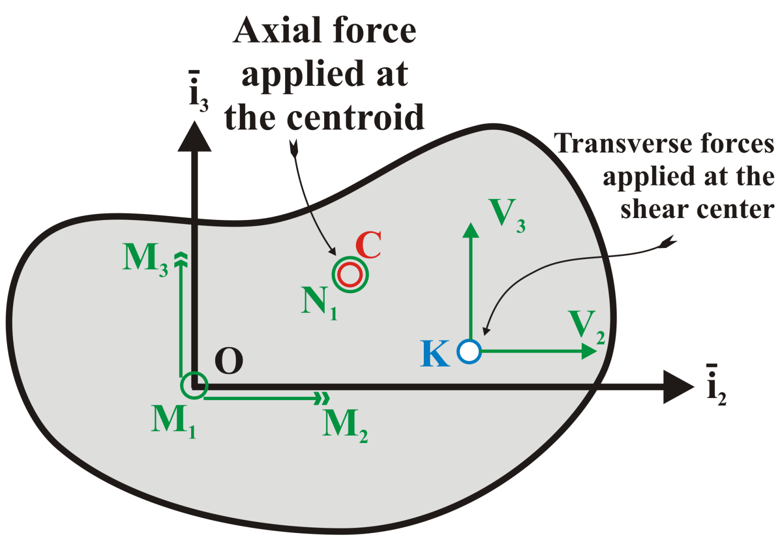

Figure 6.3: Externally applied force and moment components acting on the cross-section. Transverse shear forces are applied at the shear center, axial force at the centroid.

6. *Applied at Point.* It is sometimes convenient to be able to apply the loads at an arbitrary point of the cross-section. The `Applied at Point` button allows the definition of the coordinates (x_{2a}, x_{3a}) of point **A**, the point of application of the force vector. This option is depicted in fig. 6.4.

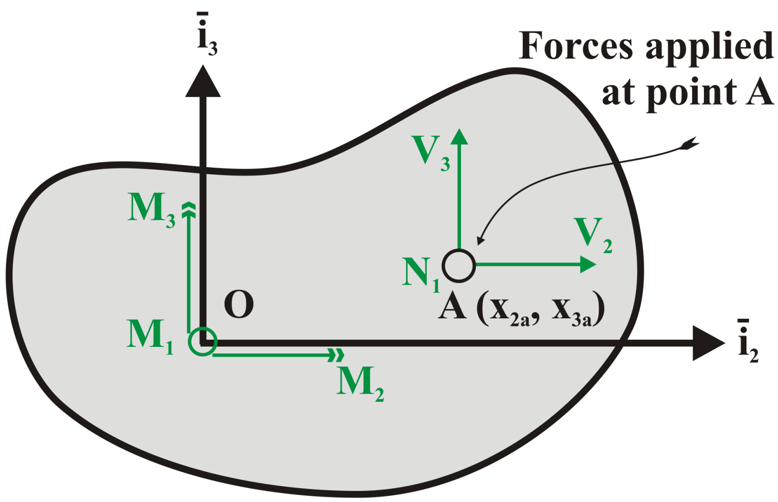

Figure 6.4: Externally applied force and moment components acting on the cross-section. The forces are applied at point **A**, with coordinates $(x_{2a},\, x_{3a})$.

Chapter 7

Geometric elements

7.1 Definition of fixed frames

A *fixed frame* consists of an origin point and an orientation triad that do not vary with time, as depicted in fig. 7.2. A default *fixed frame* called `FXDFRAME_INERTIAL` is predefined an can be used without being explicitly defined. The `FXDFRAME_INERTIAL` frame coincides with the structural axes, $\mathcal{I}$. A *fixed frame* is defined by a single dialog window, the `Fixed frame` dialog window.

7.2 The Fixed frame dialog window

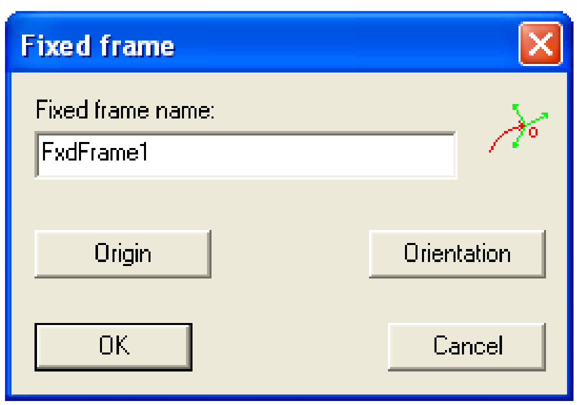

Figure 7.1: The Fixed frame dialog window.

The `Fixed frame` dialog window defines the *fixed frame* with the following data.

1. **Fixed frame name.** Enter a unique name for the *fixed frame*.
2. **Origin.** The location of the *fixed frame* is defined by the components of the position vector (x_1, x_2, x_3) of its origin measured is $\mathcal{I}$, see fig. 7.2. Note that x_1 must be zero.
3. **Orientation** The orientation of the fixed frame is defined by angle ϕ, as shown in fig. 7.2. Angle ϕ is measured in degrees and is positive in the counterclockwise direction.

7.2.1 Formatted input

The data that defines the fixed frame as described in the above section will be saved in a specially formatted input file which has the following structure.

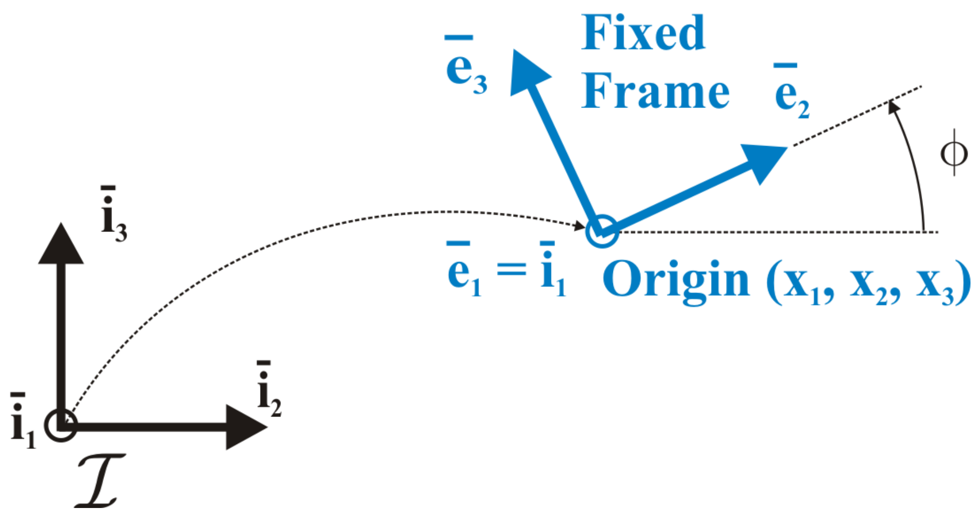

Figure 7.2: Definition of a fixed frame

@FIXED_FRAME_DEFINITION {

@FIXED_FRAME_NAME { `FxdFrameName` **} {**

@ORIGIN { x_1, x_2, x_3 **}**
@ROTATION_ANGLE { ϕ **}**

}

}

Chapter 8

Utility objects

8.1 Definition of the Include command

An *Include* command defines a list of files to be included in the definition of a cross-section. An *Include* command is defined by a single dialog window, the `Include command` dialog window.

8.2 The *Include command* dialog window

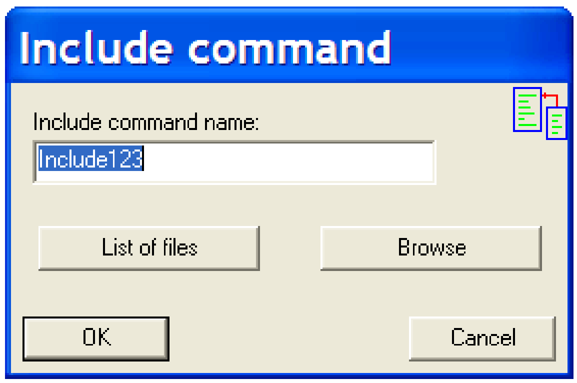

Figure 8.1: The *Include command* dialog window.

The *Include command* dialog window defines the *Include* command with the following data

1. **Include command name.** Enter a unique name for the *Include* command.
2. **List of files.** Enter the complete path and file name of the specific file to be included in the model, as shown in fig. 8.2.
3. **Browse.** Browse your computer to select the desired file, as shown in fig. 8.3. The way of selecting a file is preferable to spelling out the entire path and file name.

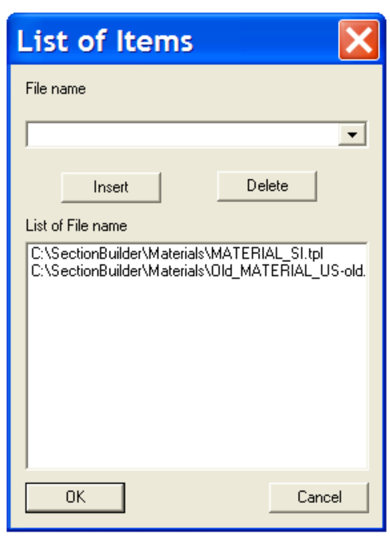

Figure 8.2: The *List of files* dialog window.

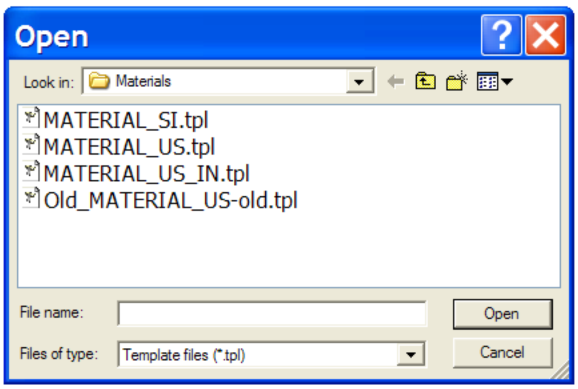

Figure 8.3: The *Browse* dialog window.

8.2.1 Formatted input

The data that defines the include command as described in the above section will be saved in a specially formatted input file which has the following structure.

@INCLUDE_COMMAND {

@INCLUDE_COMMAND_NAME { `IncludeName` **} {**

@LIST_OF_FILE_NAMES { `FileName1, FileName2,... FileNameN` **}**

}

}

8.3 Options

A number of *Options* can be selected to affect SectionBuilder operations. The following can be selected: the way in which cross-sectional shapes are allowed to be defined, the *unit system* to be used, and the *installation path* of SectionBuilder on the computer.

8.4 The *Options* dialog window

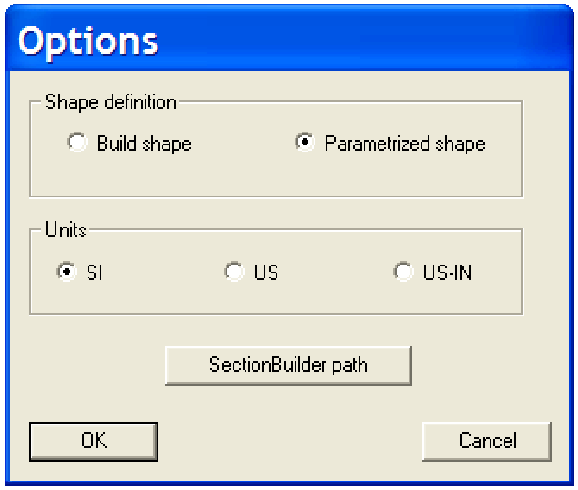

Figure 8.4: The *Options* dialog window.

The *Options* dialog window defines the following options.

1. **Shape definition.** Select *Build shape* or *parametrized shape*. The parametrized shapes are those defined in chapter 2, whereas the build shapes refer to the custom shapes discussed in chapter 3. Note that the *Build shape* option is not available under the academic license.

2. **Unit system.** SectionBuilder is not aware of any unit system. The **user must input all data in a consistent set of units**. However, SectionBuilder will provide *unit labels* in all output files according to the declared unit system. The following unit systems are available.

   (a) `SI`: international system of units.

   (b) `US`: US customary system of units.

   (c) `US-IN`: US customary system of units, but **inches** are used instead of **feet** as the unit of length.

3. **SectionBuilder path.** Enter here the complete path to the installation directory for SectionBuilder; typically, the path is C:\SectionBuilder.

8.5 Definition of Sensors

It is important to obtain numerical information about the displacement, stress or strains in the cross section when subjected to a given sectional loads. This is achieved by defining sensors at specific location of the cross-section. Sensor can be defined in two manners. Sensors can be defined more expeditiously during the visualization phase of the analysis, see section 8.6.

8.6 The Sensor dialog window

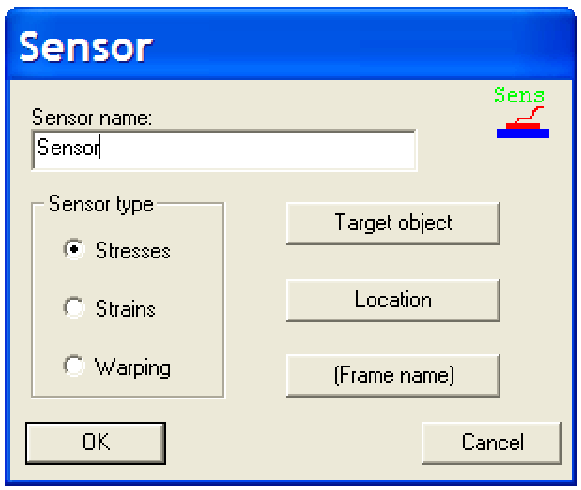

Figure 8.5: The *Sensor* dialog window.

The *Sensor* dialog window defines the sensor with the following data

1. **Sensor name.** Enter a unique name for the sensor.
2. **Sensor type.** Select *Stresses*, *Strains* or *Warping* for the sensor to compute the six stress components, the six strain components, or the three warping components, respectively.
3. **Target object.** To locate the sensor at a point of the cross-section, it is necessary to specify the solid element on which the sensor is located, then its exact position within this solid element. By defining the sensor in the visualization phase, see section 8.6, it is not necessary to provide this information. Enter the name of the solid element on which the sensor will be located.
4. **Location.** Enter the coordinates of the location of the sensor within the solid element. Here again, by defining the sensor in the visualization phase, see section 8.6, it is not necessary to provide this information.

Bibliography

[1] I.H. Abbott and A.E. Von Doenhoff. *Theory of Wing Sections*. Dover Publications, Inc., New York, 1959.

[2] G.E. Farin. *Curves and Surfaces for Computer Aided Geometric Design*. Academic Press, Inc., Boston, third edition, 1992.

[3] L. Piegl and W. Tiller. *The Nurbs Book*. Springer-Verlag, Berlin, New Jersey, second edition, 1997.

[4] R.M. Jones. *Mechanics of Composite Materials*. Taylor & Francis, Philadelphia, second edition, 1999.

[5] R.M. Christensen. *Mechanics of Composite Materials*. John Wiley & Sons, New York, 1979.

[6] S.W. Tsai and H.T. Hahn. *Introduction to Composite Materials*. Technomic Publishing Co., Inc., Westport, CT, 1980.